Program of three-dimensional seismic data processing and interpretation (surface, borehole and marine variations)

ZONDST3D

User Manual

Zond geophysical software
Saint-Petersburg 2001-2017

Contents

Program functionality *4*

Elastic waves velocity *6*

Program installation and removal *9*

Program registration **10**

System requirements **10**

Measurement units **10**

Seismogram picking module *11*

Getting started with the module **11**

Seismograms picking toolbar **12**

Seismogram picking window functions menu **14**

File reading and processing general setting dialog **18**

Survey type setting dialog **20**

Survey geometry input **21**

Cable configuration dialog (seismogram sorting) **22**

Seismogram visualization and scaling settings **24**

Seismogram scale setting dialog **24**

Traces graphic setting dialog **26**

Seismogram view mode **27**

Seismogram picking mode main options **28**

Multitab section options **30**

Operations with seismograms **34**

First arrivals picking **35**

Data filtering dialog **37**

The main stages of operating **38**

Field data inversion module *39*

Data file creation and opening for inversion **39**

Setting and modifying tomography profiles and data **41**

Setting dialog for the start model **43**

Program main window **46**

The main window toolbar **47**

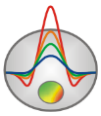
Functions of the main window menu **47**

“Hot” keys **50**

Status bar **50**

Modeling *50*

Seismic tomography forward problem solution **50**



Synthetic survey configuration creation 52
Main data file format *54*
I section of data file: observed data 54
II section of data file: tomography data 55
Features of operating with various data types in ZondST3D *55*
Cross-borehole tomography data 55
Marine survey data 56
Apparent parameters visualization *56*
Graphs plan 57
Pseudosection 57
Contour plan 58
Data inversion *60*
Inversion parameters setting dialog 60
Matching mode anisotropy 66
Attenuation tomography mode 66
Model section visualization modes and parameters *68*
Option «Buffer» in the main program menu *69*
3D model parameters setting dialog *70*
Model editor 75
Operating with model 75
Cell parameters setting dialog 76
Saving interpretation results *76*
Data import and export *77*
Usage of graphical base layers for 2D and 3D models 77
Export image setting dialog 78
Borehole data creation 79
Lithology and logging data file format *81*
Appendix 1. Palette settings *84*
Appendix 2: Graphs set editor *84*
Appendix 3: Graphs editor *85*
Appendix 4: Legend editor *87*
Appendix 5: Pseudosection parameters setup dialog *88*
Appendix 6: Axes editor *90*
Appendix 7: Model parameters setup dialog *92*
Appendix 8: 3D model editor *93*
Appendix 9: Print preview dialog *94*

Program functionality

Program **ZONDST3D** is intended for two-dimensional processing and interpretation of seismic prospecting data based on refracted waves and correlation method of refracted waves (RWCM) in surface, borehole, cross-borehole and marine variations.

ZondST3d is a comprehensive solution for seismic tomography that solves a wide range of problems from mathematical modeling and sensitivity analysis to the field data processing and interpretation. User-friendly interface and powerful functionality of data visualizing allow solving a geological problem with high efficiency.

Program is divided into two main modules. First is intended for first motion picking on seismograms. Second - for forward and inverse seismotomography problems solving.

Special interface (created for maximum simplicity and autonomous first arrivals picking) was designed for seismograms processing. The emphasis is on visualizations variety and often used functions availability. User can simultaneously pick the following types of waves: Vs, Vp. Materials processing is optimized for sharing refracted P- and S-waves.

When ray trace forward problem is being solved, a special algorithm (Shortest path's method) is used. This method allows calculating the shortest path of the refracted wave. The given algorithm has high speed of calculations and controllable accuracy.

The direct problem, i.e. ray tracing algorithm, is implemented in three ways:

1. The ray approximation. Constant velocity inside a cell.

2. The ray approximation. The linear velocity measurement inside a cell. The rates are set in nodes.

Seismotomography module allows obtaining velocity sections, both P- and S-waves, which is necessary for seismic zoning. The sources and the receivers can be located on the surface in boreholes or at the water reservoir bottom or at its surface. You can use the module for the interpretation of vertical seismic profiling (VSP). There is also an algorithm of velocities anisotropy distribution and attenuation parameter recovery. The program uses the simplest version of the anisotropy factor of seismic velocities - the ratio between Vxy and Vz. Accounting for anisotropy is very important, especially when interpreting cross-well measurements.

The "Amplitude inversion" module uses the "Attenuation tomography" algorithm and allows receiving attenuation parameter Q section in an environment of the first arrival

amplitudes values. The problem is solved on the basis of previously obtained velocity section. The amplitude values of the first arrivals are picked in parallel with the times.

An important step before field measurements is the mathematical modelling for the survey area velocity structure. Modelling allows you to evaluate the resolution and select optimal parameters for the geological problem solution.

Since the main aim of ZondST3d is velocity section parameters recovery, the program has several options for solving the inverse problem, the most important of which are: smoothing inversion – to recover smooth one, blocking for block one and focusing – to recover piecewise smooth distribution of velocity parameters with depth.

During the development of the program the possibility of accounting for a priori information was very important. Because of the equivalence of inverse geophysical problems, the results quality depends on the number of a priori settings. In **ZondST3d** there is a possibility to set measurement weights, to fix and set limits of individual cells properties range and to use a priori model as a reference during the inversion. In addition, robust schemes of the noise component assessment are used in the program. Also there is a possibility to import and display the measurement results of other methods and borehole data, which facilitates more integrated approach to the data interpretation.

For inverse problem (inversion) solution Newton's method with regularization is used. Regularization improves solution stability and allows recovering more smooth velocity distribution in the medium.

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

where A – a partial derivative matrix of measured values with respect to section parameters (Jacobian), C – the smoothing operator, W – a matrix of measurements relative errors, m – section parameters vector, μ – regularization parameter, Δf – the discrepancy vector between observed and calculated values, R – the focusing operator.

When inverse problem solving, the special attention is given to the priori information (separate measurements weights, parameters variation ranges).

ZondST3d has very powerful system of profile data visualization, the measurement editor and method sensitivity and resolution analysis system.

ZondST3d uses simple and clear data file format which allows combining various survey layout in one area. The program allows importing and displaying measurement results obtained by other methods that provides more complex approach to data interpretation.

The **ZondST3d** is a convenient program for automatic and interactive seismotomographic data interpretation and it can be used on IBM PC compatible personal computers running Windows operating system.

Elastic waves velocity

According to the elasticity theory, seismic waves velocities v_p and v_s depend on medium density ρ and its elastic modules E and σ.

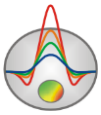
$$v_p = \sqrt{\frac{E}{\rho}\frac{1-\sigma}{(1+\sigma)(1-2\sigma)}} \qquad v_s = \sqrt{\frac{E}{\rho}\frac{1}{2(1+\sigma)}}$$

In rocks these parameters are defined by a large number of different factors – lithologic and granulometric content, cavitation (porosity, cavernosity, fracturing), fluid saturation, intrastratal pressure, metamorphism and tectonic action, occurrence depth, age, temperature etc.

Density influence on velocity variation is smallish at all, because for the majority of rocks the density variation range is limited from 1.4 to 3.2g/cm^3. Minimal relative ρ changes are typical for igneous rock, maximal – for sedimental but it's not more than 20-30%. Inverse dependence velocity from density results from the structure of the above formulas: if density relatively increase (decrease) on $\varepsilon\%$ and elastic modules values stay permanent, then velocity relatively decrease (increase) on $\varepsilon/2\%$.

According to experimental data, for the majority of rocks the positive correlation between their densities and seismic velocities v_p and v_s is typical, that is more dense rocks are usually also imply higher velocity. Such situation seems to contradict considered formulas, however, this disagreement is apparent. With increasing rock density ρ, as a rule, its Young's module E increases even faster and as a result the velocities v_p and v_s increase. The Young's module E changes to some degree in various rocks - from level about 10^2 MPa in poor sedimentary rocks to 10^5 MPa (106 kg/sm^2) in hard igneous rocks. Thus, seismic velocities values can differ in tens of times.

Poisson's ratio σ changes theoretically in the 0-0.5 range. Elastic properties of hard crystal rocks come nearer to the left limit, to the right – soft plastic rocks. In liquid incompressible mediums where there are no shift deformations σ = 0.5. In the majority of rocks Poisson's ratio has values from 0.15 to 0.35, i.e. varies relatively average value 0.25 within only ±0.1. However, in the defining v_p and v_s formulas weak σ value variations have a big influence on velocity values.



An impact of many distinctive factors on rocks elastic properties leads to an absence of unequivocal connection between geological rock definition and its velocity characteristic: absolutely different rocks can have identical velocity values, and rocks of a one name can differ by their velocities. Therefore, for various rocks or their complexes it is possible to specify only limits of probable velocity values.

P-waves velocity in rocks of uppermost disintegrated section part does not usually exceed 1 km/s. In a primary terrigenous deposits it rarely exceed 4 km/s and in carbonate and hydrochemical rocks it could increase to 6 km/s. In igneous and metamorphic rocks v_p reaches the value of 6.5-7 km/s, generally increases with it basicity growth and metamorphism rate. Near the bottom of the Earth's crust (Mohorovičič discontinuity) $v_p$value reaches the value of 8 km/s, judging by refracted wave velocity. In truth, material composition of rocks at this depth is not reliably known.

S-wave velocities generally depend on velocities of P-waves. Theoretical ratio is $\gamma = {}^{v_s}/_{v_p}$ and can vary within limits from 0 (fluids) to $1/\sqrt{2} = 0.7$ (if σ = 0). For the majority of consolidated rocks γ value is between 0.4 and 0.6, that matches to σ values range 0.4-0.2. But in very loose argillo-arenaceous rocks value γ can be about 0.2-0.3 and even less. In general, v_s distribution is studied worse than those for v_p.

Rocks with the same lithologic composition may considerably differ by seismic waves because of internal structure determined by forming conditions. It is especially noticeable in terrigenous rocks formed in different hydrodynamic conditions: sediments formed in low-energy environment of a shelf and granulometrically more homogeneous have lower velocities than rocks accumulated in high-energy littoral environment and granulometrically less sorted. Relatively low-velocity rocks are organogenic carbonates as compared to chemogenic.

Rock cavity degree is a significant factor for seismic waves velocities, first of all, its porosity. All other conditions being equal, with the increasing of rocks porosity seismic velocity decreases. This dependence is the most obvious in terrigenous deposits where the porosity can reach the values of 30-40 %. These rocks such as sand and sandstones are considered as the heterogeneous media consisting of a hard skeleton (grains) and pore space filled by fluid (liquid, gas and their mix). For similar model v_p velocity is estimated by the approximate empirical ratio known as the equation of average time:

$$\frac{1}{v_p} = \frac{k_{\text{п}}}{v_{\text{ф}}} + \frac{1-k_{\text{п}}}{v_{\text{т}}},$$

where k_p - porosity factor, v_t and v_f – P-waves velocities in a hard skeleton and a fluid respectively.

Dependence $v_p(k_p)$ has an exponential character. Gas presence in waterlogged rocks leads to decrease in rocks general elasticity and in velocity in it, as a result of high compressibility. When pores are fully saturated with water, there is a sharp decrease in the rocks compressibility and abrupt increase of v_p velocity. Degree of water saturation has almost no effect on S-waves velocity v_s. This effect is used to determine the seismic groundwater level (GWL) in unconsolidated sediments.

Water freezing located in pores, caverns, cracks causes sharp increase of seismic velocities in rock, because v_p velocity in ice is almost 2.5 times more than in water. Depending on porosity, relative content of frozen and not frozen water, its mineralization and temperature, P-waves velocity can increase on 1-2 km/s. It essentially changes velocity characteristics of the top section part in the zones of frozen rocks spreading: so-called velocities inversion is observed here when from the surface on some depths interval velocities decrease while it is changing from frozen to thawed deposits.

For sedimentary rocks more or less essential increase in velocity with depth is typical because of increasing of mountain pressure. On the one hand, it is due to porosity decreasing, what increases v_p, according to (5), since $v_t > v_f$. From the other hand, v_t velocity depends on the contact area of adjoining rock grains that increase with increasing pressure. As result Young's module and velocities v_t and v_p increase. Velocities increasing with depth attributable to terrigenous rocks which differs by high initial porosity. For less porous carbonate deposits this property appears much more poorly and for chemogenic rocks it is almost indistinguishable.

With identical material composition and an equal depth more ancient rocks differ in higher velocities. Such effect is due to the duration of diagenesis, metamorphism and tectonic stress processes impact which makes rocks more rigid and elastic. This dependence is quite weak: velocity increases approximately as a sextic radical from absolute age of deposits.

In porous deposits on considerable depths velocity depends on a ratio of external geostatic pressure from overlying rocks to the intrastratal pressure of the fluid filling pores. The first one increases velocity reducing porosity. The second one prevents porosity reduction and decreases velocity creating thrust in rock grains. The effective pressure which defines velocity value is almost equal to a difference of external and internal pressure. In some cases fluid outflow from sandy deposits under the influence of geostatic pressure is hindered by screening effect of clay rock. For this reason intrastratal pressure is greater than normal hydrostatic pressure at this depth, and v_p velocity in the layer is below the normal level.

Zones of tectonic disruptions are characterized with significant reduction of seismic velocities which can reach 30-40 % and more, depending on rocks disintegration degree. This effect is usually considerably higher for S-waves therefore the velocities ratio $\gamma = {}^{v_s}/_{v_p}$ in zones of tectonic violation, deconsolidation and cleavage of rocks has a lower value.

When the lithology composition of sedimentary rock stays permanent, the lateral velocity changes are observed which are due to plicate structural forms. Velocity reduction is more often noted in the crest of reservoir where the rocks cleavage is more developed. But the false effect caused by increased dynamometamorphism in crest of folds can also occur.

Experimental data prove that seismic velocities variations can display changes of a rock tension. In particular, velocities reduction and their anisotropy appearance can coincide with zones of deconsolidation and cleavage of rocks of tectonic origin.

Another important property which is quite often found in rocks is the anisotropy of their elastic properties, resulting in dependence of seismic velocities values on the direction. Such an effect arises due to both thin-layer deposits structure and spatially oriented rocks cleavage. According to the experimental data, among lithologically homogeneous formations clay deposits which are characterised with ordered scaly structure have the greatest anisotropy. For them k_p for P-waves reaches values as high as 1.2-1.5 and higher. In sandstones k_p rarely exceeds 1.1-1.2 values. Carbonate deposits have weak velocity anisotropy. For S-waves anisotropy is usually higher than for P-waves in the same sections.

Another reason for seismic velocities anisotropy is intensive rocks cleavage breaking the continuity of their mineral skeleton. If there are crack systems with fixed direction, waves velocity is maximum along it, and minimum perpendicular to it. The defined anisotropy factor depends on shape and the relative cracks sizes in comparison with the prevailing wave length, and also on elastic properties of cracks filler [Boganik G. N., Gurvich I.I. Seismic exploration. 2006].

Program installation and removal

The program **ZONDST3D** is released on a compact disc or on the Internet. The distribution kit includes this Manual. You can load the latest program updates on the site: www.zond-geo.ru.

To install the program, copy the program from the compact disc in the chosen directory. To update installation, just install the new program version over the old one.

Before the first program start it is necessary to install the driver of protective key SenseLock. To do this, open the SenseLock folder (the driver can be loaded from a compact disc, or on a site) and run the file InstWiz3.exe. After driver installation insert the key. If everything is done, a message that the key is found appears in the bottom system panel.

To remove program, delete the program operating catalogue.

Program registration

To register the program, press the “Registration file” item (in case you do not have the USB key) in the main menu. In the resulting dialog fill in all the fields, choose the registration file name and save it. The created file is sent to the address specified in the contract, then the user receives the unique password connected with the serial number of a hard disk which is necessary to input in the "Registration" item. The second registration option is binding with the delivered key SenseLock. Thereat it is necessary that in operating time the key is inserted into the USB port.

System requirements

The program **ZondST3d** can be installed on the computer with Windows 98 operating system and later versions. Recommended system parameters: P IV-2 GHz processor, 512 Mb memory, a display resolution 1024 X 768, colour mode - True colour. (Display resolution should not be changed during working with data).

As the program nowadays actively uses system register resources, in systems later than Windows XP, it should be run as administrator (the right mouse button click on a program icon –run as Administrator).

Measurement units

Geometrical units (coordinates of sources and receivers, topography) – **meters**.

Units of the first arrivals times – **milliseconds**.

Units of seismic waves velocity– **kilometres per second**.

Seismogram picking module

The field data processing aim is receiving of target wave first-arrival times hodograph. To start field data processing, it is necessary to have the file or files of field measurements in SEG-Y format. Process of seismogram picking can be performed in the special module (fig. 1) which is called the **Trace editor** in the program main menu or by clicking the button on the toolbar. After running this module, it is necessary to load field seismograms into it and start picking process.

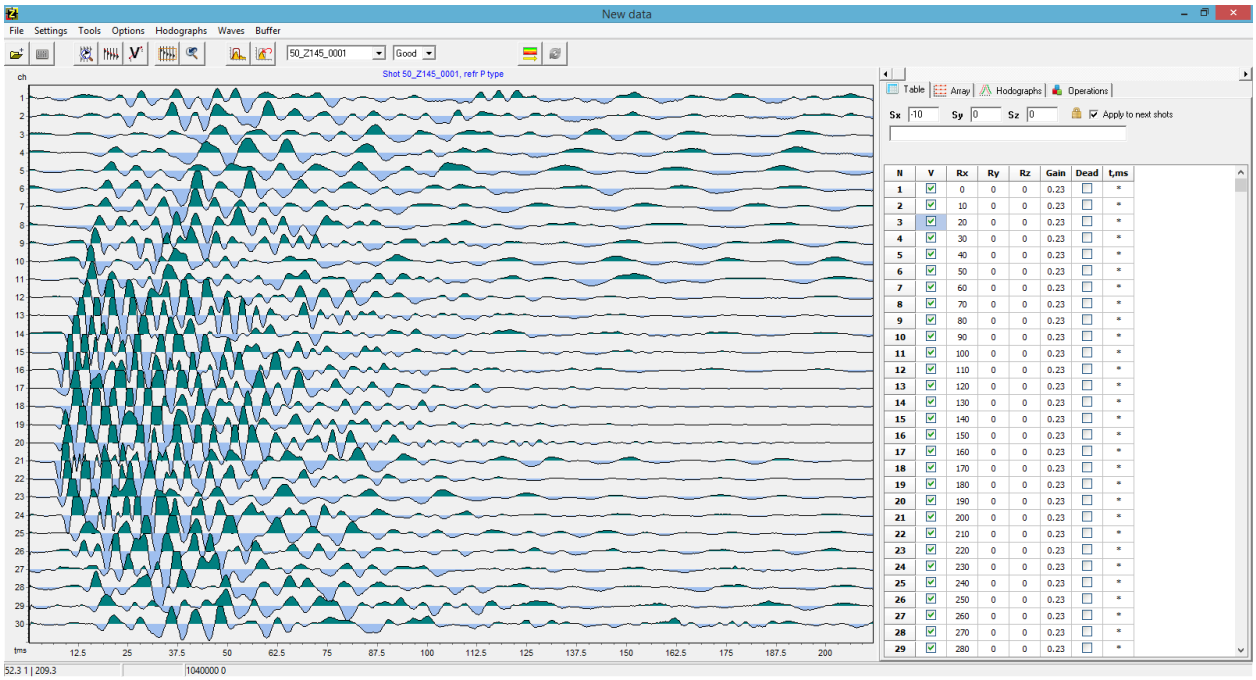
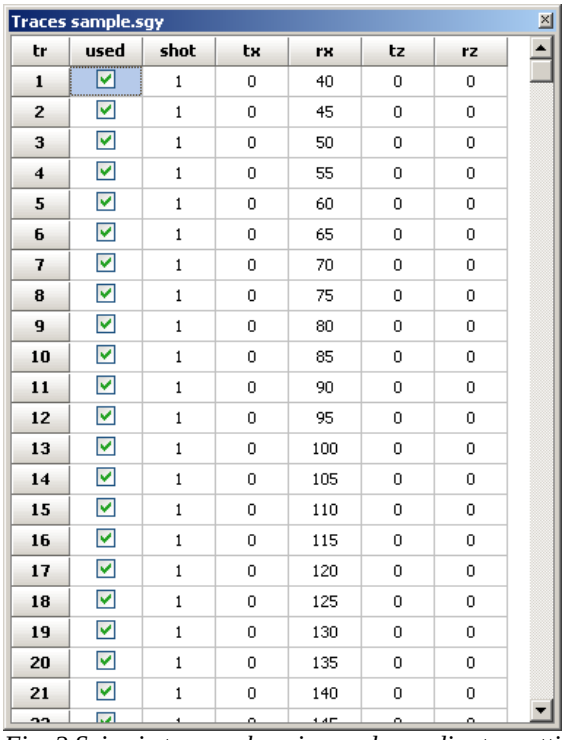


Fig. 1. Operating window of seismogram picking module Trace editor.

The window is divided into two sections: a graph of seismograms displaying (leftward), multitab section for setting and displaying survey layout parameters (rightward).

Getting started with the module

Work begins with opening one or several SEG-Y files or the project file. Once the file is loaded, the **Show parameters for every file** dialog appears. If you press the *Yes* button, there will be the window in which the user is offered to choose the necessary seismic traces and to set coordinates of sources and receivers if it is necessary (it is possible to set coordinates of receivers and sources later) (fig.2).



tr	used	shot	tx	rx	tz	rz
1	☑	1	0	40	0	0
2	☑	1	0	45	0	0
3	☑	1	0	50	0	0
4	☑	1	0	55	0	0
5	☑	1	0	60	0	0
6	☑	1	0	65	0	0
7	☑	1	0	70	0	0
8	☑	1	0	75	0	0
9	☑	1	0	80	0	0
10	☑	1	0	85	0	0
11	☑	1	0	90	0	0
12	☑	1	0	95	0	0
13	☑	1	0	100	0	0
14	☑	1	0	105	0	0
15	☑	1	0	110	0	0
16	☑	1	0	115	0	0
17	☑	1	0	120	0	0
18	☑	1	0	125	0	0
19	☑	1	0	130	0	0
20	☑	1	0	135	0	0
21	☑	1	0	140	0	0

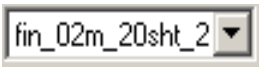
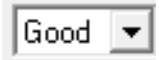
Fig. 2 Seismic traces choosing and coordinates setting window

Before loading SEG-Y files, make sure that the file reading settings **Settings/File settings** are correct (more information in the**File reading and processing general setting dialog** section).

Seismograms picking toolbar

The toolbar serves for quick calling the most often used functions in the module. It contains the following buttons (from left to right):

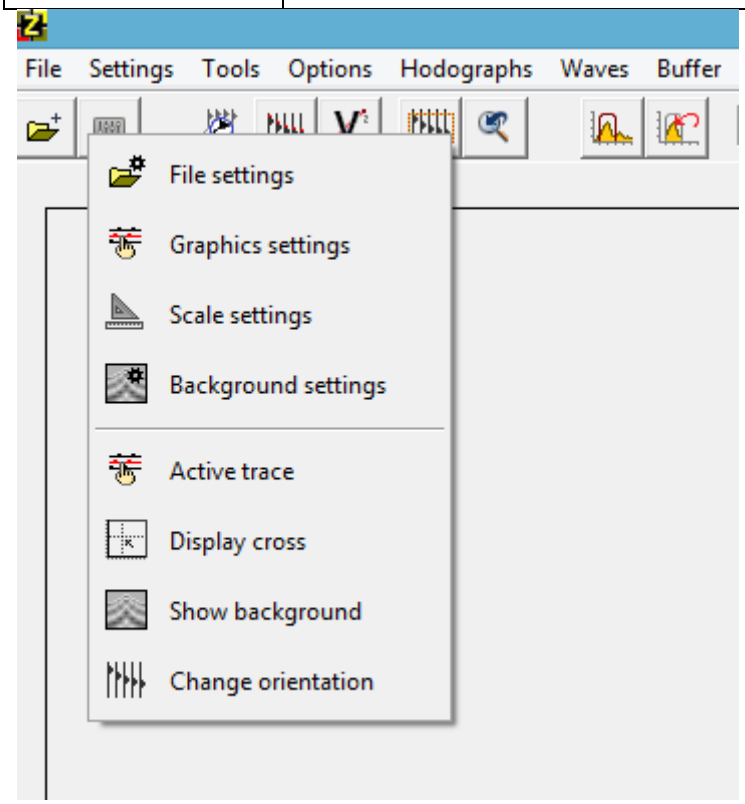
	Open field data file or files in SEG-Y format or operating project.
	Pop-up menu to run various dialogues and settings.
	Activate zoom/selecting mode for seismogram section. Selecting is performed through rubber rectangle using a mouse. Once this mode is activated, two following buttons appear.
	Set the operating data window. Window size is set in the mode. Operating window is used when filtering and auto scaling seismic traces.
	Move to the previous seismogram selecting section.
	Activate the first-arrivals picking mode. The choice of the first arrivals is

	performed through the left mouse button. Right-clicking deactivates the chosen value on a hodograph. The program interpolates values of the first arrivals between the positions of the first and the second mouse clicking. Once this mode is activated, two following buttons appear.
	The Multipicking option. The first arrival times are set automatically from the first to the last selected point on the seismograms.
	The Autopicking mode. Activate precise first arrival times positioning mode. New value is defined depending on the method set in the correction settings (tie to zero, to the extreme, etc.). To change the settings, right-click on the button.
	Activate the mode of velocity determination between two points. To determinate velocity in the chosen seismogram section, it is necessary to choose the first point position. Then, while pressing the mouse button, move cursor to the necessary position. Velocity value will be displayed in the bottom of the status panel (the second section). Before determining velocity make sure that coordinates of receivers are set correctly.
	Run data filtration dialog.
	Cancel filtration results.
	Choose an active seismogram from the list. There are all the project seismograms in the list.
	Seismogram quality assessment.
	Apply changes and move to the data modeling and inversion window.
	Refresh data and go to data modelling and inversion window.

General file reading settings dialog contains the following functional buttons:

	Set the parameters of file reading.
	Run the seismic traces graphical settings dialog.
	Run the image scale setup dialog.
	Run the background parameters settings dialog.
	Activate the selection of active trace mode through the mouse. The active trace is highlighted in green colour (default) and is drawn last. This mode should not be used during picking, because it slows down the program work.

	Show the crossing lines following to the cursor.
	Display background showing a seismogram in the interpolation form.
	Flip seismic traces through 90 degrees. Depending on data type, it is convenient to use vertical or horizontal seismogram orientation.



Seismogram picking window functions menu

The names of menu items and their aim are listed below:

File	Open SEG-Y/Project	Open file, files set of field data in SEG-Y format or working project.
File	Add to project	Add data to the operating project. This option serves to join data in a single project.
File	Save project	Save correlation data and results in the operating project.
File	Close project	Close all the seismograms.
File	Print preview	Run the image printing dialog.
Settings	Survey type	Field geometry parameters settings dialog.
Settings	Set working area	Set data operating window. Window sizes are set in the mode. Working window is used for seismic traces autoscaling. If in the multitab section of **Table** tab the option *Apply for next shots* is activated, the operating window is assigned for

		all the subsequent project seismograms.
Settings	Change orientation	Flip seismic traces through ninety degrees. Depending on the data type is convenient to use a vertical or horizontal seismogram orientation.
Settings	Background settings	Run the interpolating background graphic settings dialog.
Settings	Show background	Show background displaying a seismogram in the interpolation form.
Settings	Background color	Run seismogram background fill settings dialog.
Settings	File settings	Run file general reading and processing settings dialog.
Settings	Graphic settings	Run graphs settings dialog.
Settings	Scale settings	Run image scale settings dialog.
Settings	Units	Specifies how to display data: by channels, by distance, by offsets, in meters, milliseconds, or counts.
Settings	Set sample time	Set sample interval, miliseconds
Tools	Set working area	Set data operating window. Window sizes are set in the mode. Working window is used for seismic traces autoscaling. If in the multitab section of **Table** tab the option *Apply for next shots* is activated, the operating window is assigned for all the subsequent project seismograms.
Tools	Select all	Set whole seismogram as a data operating window.
Tools	Filtering	Run data filtering dialog.
Tools	Undo filtering	Return to the initial data.
Tools	Picking mode	Activate the first-arrivals picking mode. The choice of the first arrivals is performed through the left mouse button. Right-clicking deactivates the chosen value on a hodograph. The program interpolates values of the first arrivals between the positions of the first and the second mouse clicking.
Tools	Zoom mode	Activate zoom/selecting mode for seismogram section. Selecting is performed through rubber rectangle using a mouse. Then you can activate the autoscaling mode for the chosen window.
Tools	Display cross	Display the crossing lines following the cursor.
Tools	Edit source pos	Run the dialog of a fast source coordinate defining for the list

		of files.
Tools	Set cable configuration	A very important dialog allowing areal seismogram sorting for the more convenient picking. Hodographs will also be displayed in accordance with the chosen order.
Options	Delete current shot	Remove the current seismogram from the project.
Options	Delete empty shots	Remove all the seismograms for which the picking has not been performed from the project.
Options	Delete dead traces	Remove the "dead" traces. The traces which are selected in the column Table of the Dead tab in the multitab section will be removed. In this column the user can mark the faulty traces.
Options	Sort shots	Sort seismograms by the source position.
Options	Combine shots	Add seismograms, selected in the Operations tab of multitab section in the Shot field, to the current seismogram. It is convenient when using short cables. In this case, as a rule, a cable can be moved along the profile. This option allows getting a long record from one source for several positions of receivers. Before use, check the correctness of the coordinates recording.
Options	Fix current shot	Fix (save) the current trace as an original with all the processing details.
Options	Apply AGC	Apply AGC (Automatic Gain Control). The option performs record normalizing by the average value of its level in a certain registration time interval (window). The window size is set in samples and selected experimentally. To cancel the action use the button of canceling data filtering
Options	Remove aver in window	Remove the average value in the specified window. It allows you to select the high-frequency signals. To cancel the action use the button of canceling data filtering
Options	Collect amplitudes	Collection of the first arrivals amplitudes where the time

	shot	picking is performed for the seismogram. This option allows collecting amplitudes not at the picking point, because it may be zero, but in a window around it. The window size is specified in samples in the File settings/Amplitudes picking menu.
Options	Collect amplitudes all	Collection of the first arrivals amplitudes where the time picking is held for all the seismograms.
Options	Undo action	Undo the last picking action.
Hodographs	Delete current	Remove the current hodograph.
Hodographs	Autocorrect current	Autocorrect the current hodograph (correlation of already picked arrivals based on the picks character near which they are located).
Hodographs	Project current	Automatic picking mode based on using the two adjacent picked hodographs which will be projected onto the current seismogram.
Hodographs	Project empty times	Pick the current seismogram on the basis of the results of adjacent seismograms picking.
Hodographs	Copy current	Copy the current hodograph to the buffer.
Hodographs	Paste to current	Paste data from the buffer to the current hodograph.
Hodographs	Load picks	Load picking data from the file.
Hodographs	Save picks	Save the current picking results to the file. The following file types are available: *Laccolite god file* – save the current hodograph as a *.god file, *Laccolite directory* – save hodographs for all seismograms in the specified directory as *.god file, *SRT file* – save *.SRT file.
Hodographs	Smooth all	Smooth all hodographs.
Hodographs	Smooth current	Smooth the current hodograph.
Hodographs	Smooth reciprocity data	Average the reciprocal times.
Hodographs	Correct t0 (reciprocity)	Correct the record start by the reciprocal points.
Hodographs	Correct t0 (calculated)	Correct the moment marks (record delay time) on the basis of the calculated hodographs.

Hodographs	Reciprocity error	Показать среднюю ошибку корреляции данных, рассчитанную на основе принципа взаимности. Show the average data correlation error which was calculated being based on the reciprocity principle.
Waves	P- refracted	Set the picking waves type - P- refracted.
Waves	S-refracted	Set the picking waves type - S- refracted.
Waves	Exchange P&S	Поменять тип пикированных волн. Опция используется в случае, если при пикировке был установлен неверный тип волны. Change the picking waves type. The option is used when false waves type was set when picking.
Waves	Remove active data	Remove active data.
Buffer	Picks1…	Save current picking results in the buffer. It is possible to save up to 5 variants of the same hodograph. Saved variants can be loaded by clicking on them again. After second clicking the dialog Working with buffer appears. It contains two buttons: From buffer - load the saved hodograph instead of the current one, To buffer - save the current hodograph to the buffer.

File reading and processing general setting dialog

You can run the dialog of file reading and processing general settings by using the option **Settings/File settings** or the toolbar button .

The **SEG-Y** tab contains options managing the field data files format (fig. 3).

Before creating a new project from the SEG-Y files, it is necessary to be sure that settings of this tab correspond to the files in usage.

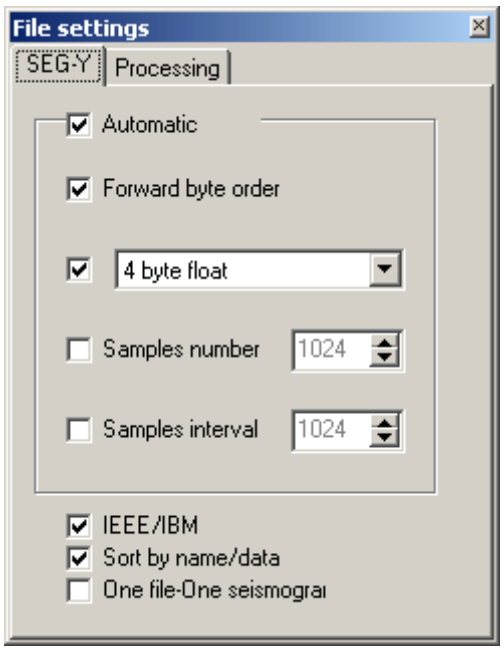


Fig. 3. ***File settings*** *dialog, the* ***SEG-Y*** *tab.*

The *Automatic* option activates the mode of automatic format detection. If this option is deactivated – the following settings are available.

The *Forward byte order* option - sets an order of bytes reading. The pop-up list is intended for a data type choice.

The *Samples number* option – sets number of counting in the seismic trace. The *Samples interval* option – sets an interval of seismic trace digitization (in samples).

The *IEEE/IBM* option – sets the PC/UNIX number format.

The *IEEE/IBM* and *Forward byte order* option are mainly used.

The *Sort by name/data* option – specifies to the program an order of seismic trace sorting– by file name or by date.

The *One file-One seismogram* option -allows determining each loaded file as a single seismogram.

The **Processing** tab includes secondary settings (fig.4).

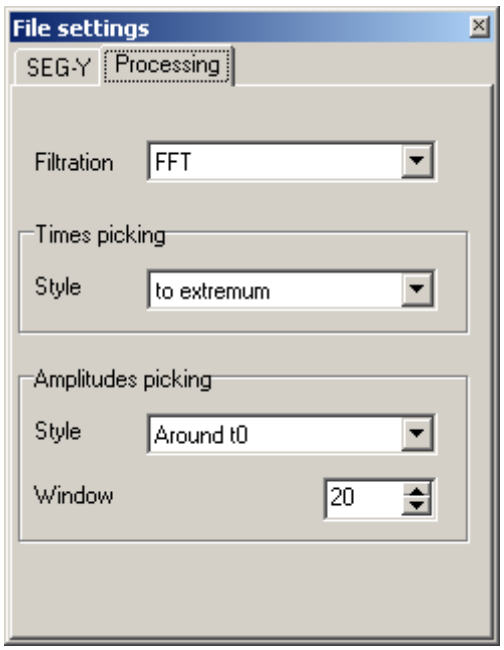


Fig. 4. ***File settings*** *dialog,* ***Processing*** *tab.*

The first pop-up list is intended for a choice of Fourier transformation algorithm (the fast - FFT or usual DFT). The second pop-up list allows choosing the type of automatic correction of the first-arrivals in the mode. *To null* value – reduce to the flex point of the seismic trace, *to maximum* – reduce to the local extremum of the seismic trace, *to minimum* – reduce to the seismic trace minimum, *to extremum* – reduce to the local extremum of the seismic trace.

The third pop-up list *Style* allows specifying amplitude picking type: about t0 or after t0. It is necessary to specify window size in the *Window*.

Survey type setting dialog

The Dialog is available in the main menu of the seismogram picking module Settings/Survey settings (Fig. 5).

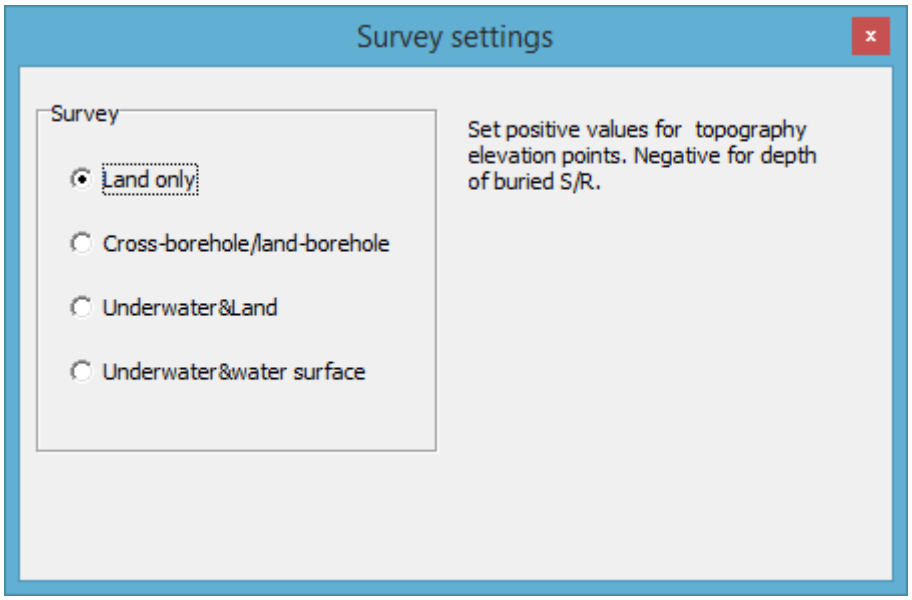


Fig. 5 Survey settings dialog

The survey settings dialog window is divided into two areas. On the left there is an area Survey in which the observations type is set:

Land only – survey configuration earth-earth. In this case, the Table tab of the multitab section will contain the topography data in the column Rz.

Cross-borehole/land-borehole – survey configuration of cross-borehole tomography/vertical seismic profiling. In this case, the Table tab of multitab section will contain depths data in the column Rz.

Underwater & Land - combined survey configuration.

Underwater & water surface - surface and underwater survey configuration.

Survey geometry input

If survey geometry has not been input when the **SEG-Y** files were being recorded, you can input the data using the program options. You can initially input the geometry using indexes (1,2,3 …) and then load the file of coordinates by option in the **Hodographs** tab of the multitab section. In the **Multitab section options** section the functions allowing setting the survey geometry fast are described. Due to the fact that the data for the same source position may contain repeated measurements, it is convenient to enter the source coordinates firstly and then sort the seismic trace by the source position. This allows estimating and removing data and, if necessary, needless records.

The **Set source positions** dialog called by the **Tools/Edit source pos** option allows you to input indexes of source positions in accumulation files list (fig. 6).

In the top row of this dialog in the *dx*, *dy* and *dz* fields the index increment for X, Y and Z is specified. The button assign an index to the following file along X equal to Sx+dx, along Y Sy+dy and along Z Sz+dz. For fast *Sx*, *Sy* and *Sz* columns filling for the same source position accumulation, it is necessary to hover the cursor over Sx of the last file, and with pressed SHIFT button click on Sx of first file relating to current source position.

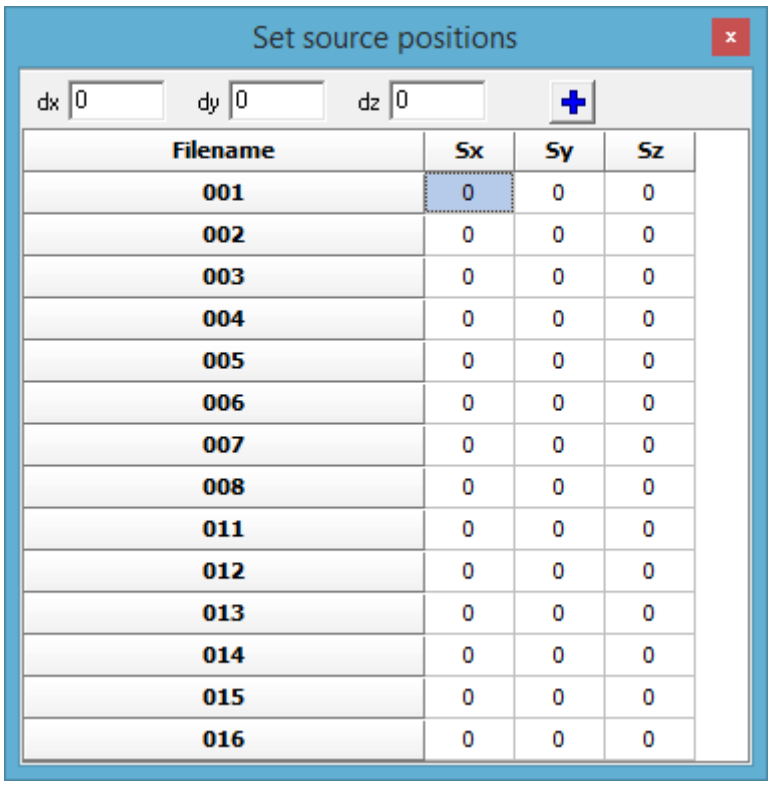
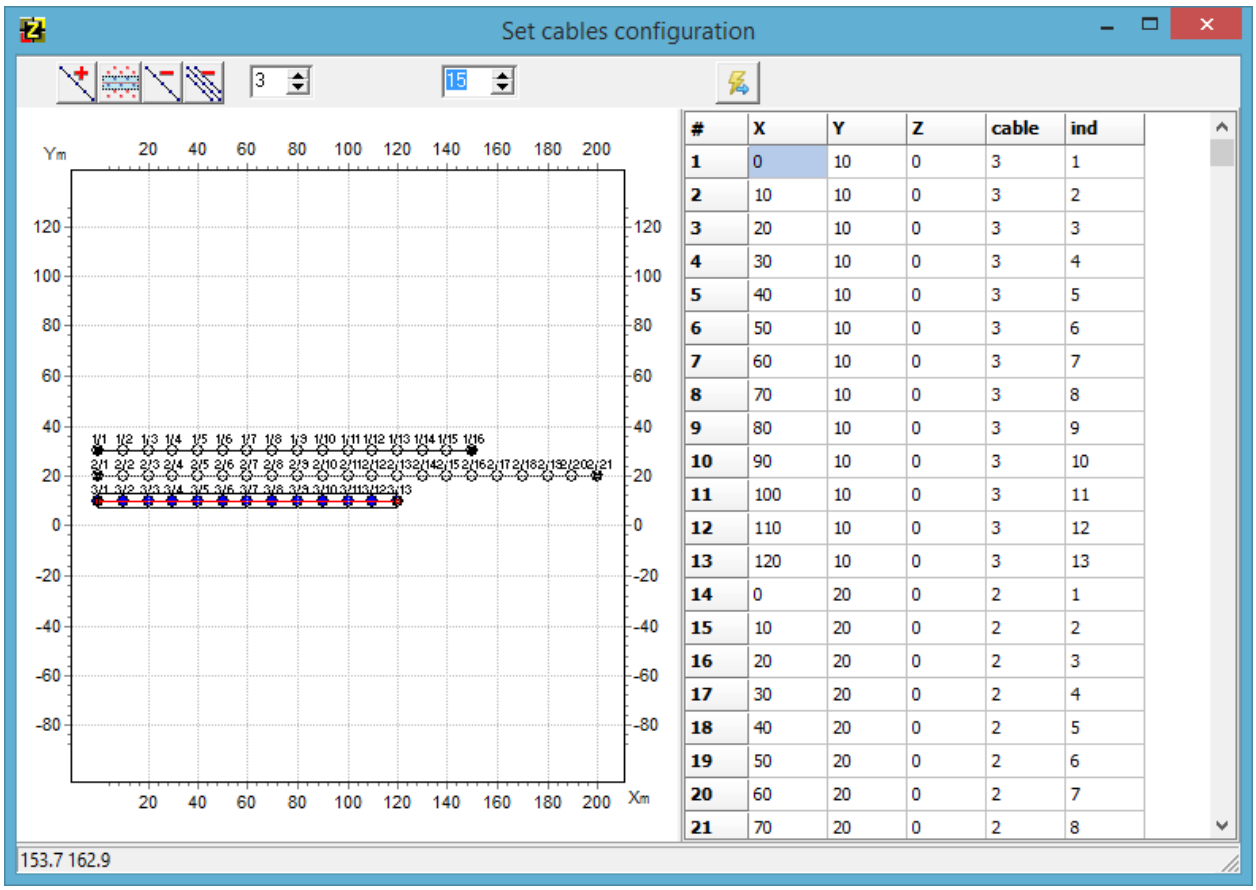


Fig. 6. ***Set source positions*** *dialog.*

Source and receiver indexes can be also input in the *Rx*, *Ry*, *Rz* and *Sx*, *Sy*, *Sz* columns in the **Table** tab of the multitab section

Cable configuration dialog (seismogram sorting)

The dialog (Cable configuration option) is an instrument for an adequate seismogram sorting and for the more convenient picking. It is an known fact that rate and convenience of picking process for areal survey depend heavily on how the traces are sorted. This dialog allows sorting the seismograms in an arbitrary, user defined way.



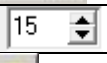
#	X	Y	Z	cable	ind
1	0	10	0	3	1
2	10	10	0	3	2
3	20	10	0	3	3
4	30	10	0	3	4
5	40	10	0	3	5
6	50	10	0	3	6
7	60	10	0	3	7
8	70	10	0	3	8
9	80	10	0	3	9
10	90	10	0	3	10
11	100	10	0	3	11
12	110	10	0	3	12
13	120	10	0	3	13
14	0	20	0	2	1
15	10	20	0	2	2
16	20	20	0	2	3
17	30	20	0	2	4
18	40	20	0	2	5
19	50	20	0	2	6
20	60	20	0	2	7
21	70	20	0	2	8

Рис. 7. Окно конфигурации кос.

In the left graph all the cable receiver positions in the chosen plane (XY, XZ, YZ) are displayed. The displaying plane is chosen in the the top toolbar pop-up window which appears in case of cross-borehole survey type.

The cables are selected one by one in the way that the traces must be sorted.

This dialog window toolbar contains the following options:

	Load a raster map file, topography or observed data as a background (fig. XXX).
	Load Google map.
	Add a profile line. To set profile lines click the left mouse button; to set the last point click the right mouse button.
	Include measurement points in profile automatically. The points contained in a rectangular area around the specified line.
	Remove current profile
	Remove all profiles.
	Set the current profile number
	Set the size of automatic receiver selection area.
	Confirm and back to the picking mode

The receiver coordinates and their new indexes which will be used for seismograms displaying are shown on the left.

Seismogram visualization and scaling settings

For the convenience of analyst work various options of scaling and visualization seismograms are provided in the picking mode.

The zoom of a certain section or its moving is performed in the selection/zoom mode (the tool – "a rubber rectangle"). For section selection which you want to zoom, the mouse cursor moves down and to the right, with the pressed left button (Fig. 8A).

The button *Set working area and autoscaling trace* allows specifying are convenient for the selected area trace scales automatically.

To move to the previous section of the seismogram selection use the option . To return to initial scale, do the the same actions, but the mouse moves up and to the left (the Fig. 8B).

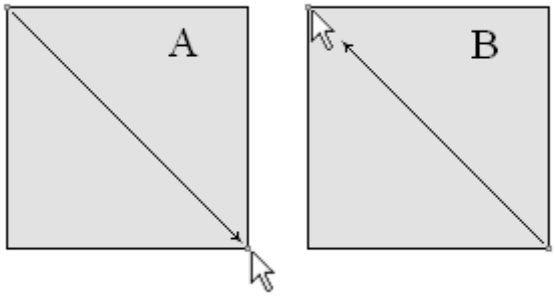


Fig. 8. The mouse moving directions for the scale changing.

Seismogram scale setting dialog

To run seismogram scales settings dialog, use the **Settings/Scale settings** option or the button on the toolbar , then press the button.

The **Scaling options** dialog contains two tabs.

The tab **Scales** contains the options determining scales of seismic traces graphs and the whole image (fig. 9).

The **Norm style** section sets normalization type for seismic traces graphs. The *Common maximum* value scales the graphs by the general seismogram maximum. The *Trace maximum* value scales every graph by its own maximum.

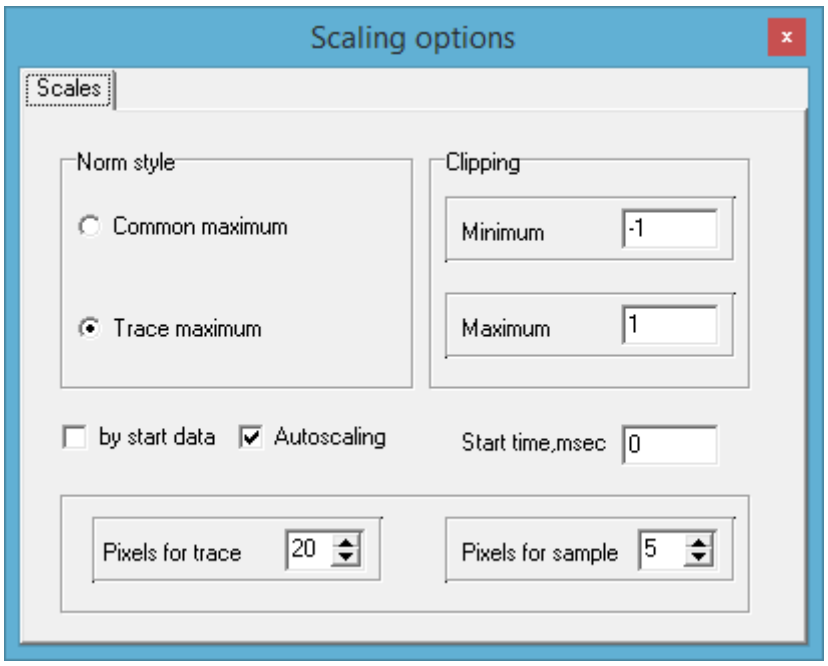


Fig. 9. ***Scaling options*** *dialog,* ***Scales*** *tab.*

The *Clipping* area sets graphs maximum and minimum limiting values on reaching which the graphs are cut off.

Maximum and *Minimum* values are set on the basis of the fact that the distance between the adjoining traces is equal to unity.

The option *By start data* – indicates to the program that graphs scaling will be performed for the initial (not filtered) data values.

The option *Autoscaling* – sets the graph image scale. If the option is activated, the scale is set automatically on the basis of the window size. If the option is deactivated – image scales are set by the options *Pixel for trace* (pixels number for one trace) and *Pixel for sample* (pixels number for one counting).

The option *Start time* sets a moment mark value (positive number in mSec or counting) for all the seismic traces. The times/counting axis is displaced so that a moment mark is equal to an axis zero. Thereat appearance of negative values on an axis is possible.

Traces graphic setting dialog

To run seismic traces graphic settings dialog, use the option **Settings/Graphics options** or the button on the toolbar , then press the button (fig. 11).

The **Trace** tab contains the options which are responsible for the appearance of seismic traces graphs. Two main **Primary** and **Additional** sections are responsible for settings of the main and additional (displaying over the main in the *Multi* mode) seismograms respectively.

The **Negative** and **Positive** sections include color settings of graphs minimum and maximum (*Fill color* – color of filling, *Transparent* – without filling).

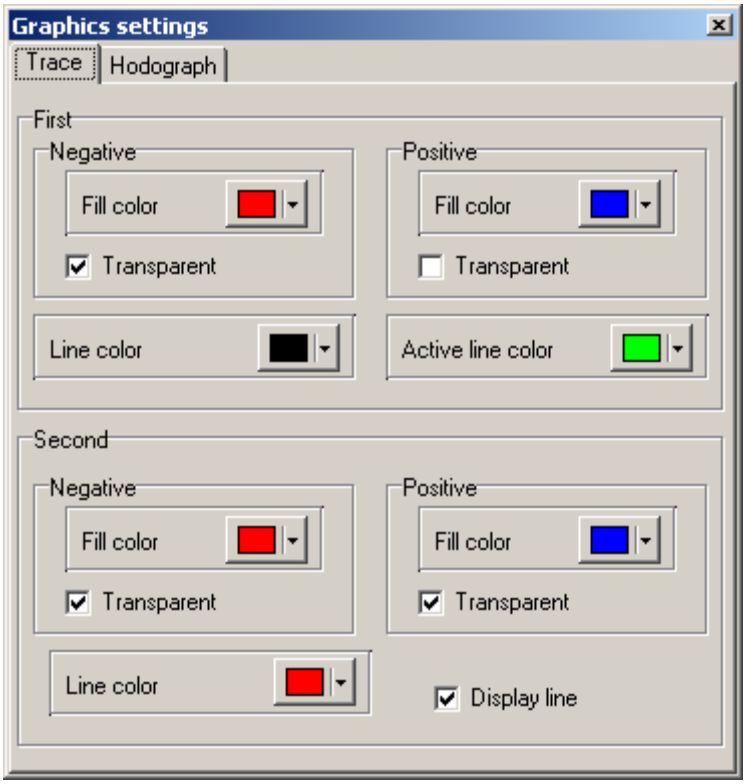


Fig.11. ***Graphics settings*** *dialog.*

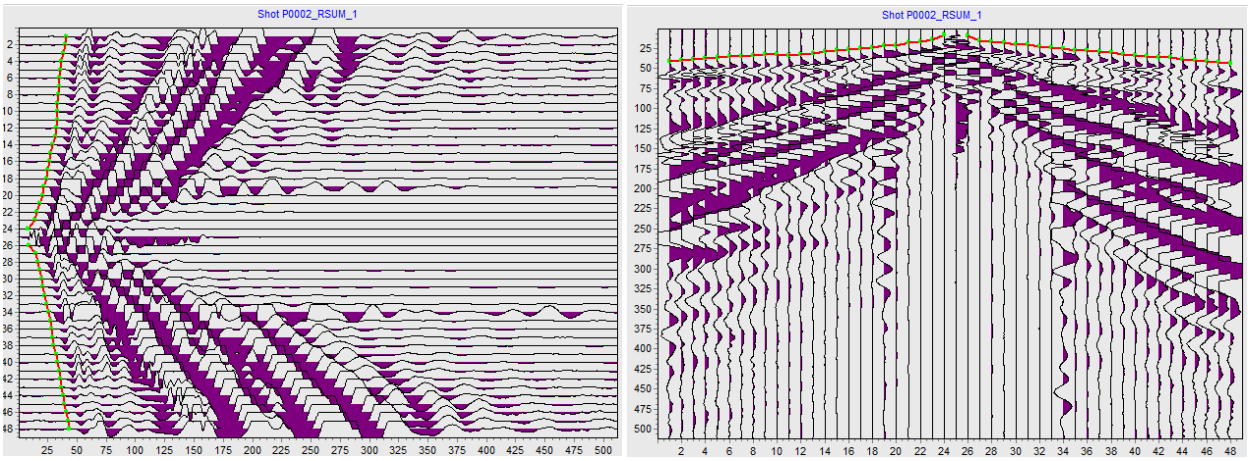
The option **Line color** sets line color of graphs. The option **Active line color** sets line color of the active graph. The option **Display line** - indicates whether it is necessary to draw graph lines.

The **Hodograph** tab contains hodograph color settings displaying on seismograms during picking. The options **Current, Previous, Next, Reciprocity, Calculated** set colors for the current, previous, following, mutual and calculated hodograph. If the option is activated – this hodograph will be displayed (if it exists).

Button [Colors] runs graphs plot editor settings dialog (more details in **Appendix 1: Graph set editor**).

Seismogram view mode

Depending on the data type it is convenient to use vertical or horizontal seismogram orientation. The option **Settings/Change orientation** allows rotating seismogram by 90° (the Fig. 12).



Puc.11. Horizontal and vertical seismogram orientation.

To display background which represents seismogram in an interpolated form use the button , than press the button or use the menu function **Settings/Show background** (Fig. 13). Color palette can be set by running **Color palette** dialog with the menu function **Settings/Background settings** (Fig. 14). (the button in the file reading dialog)

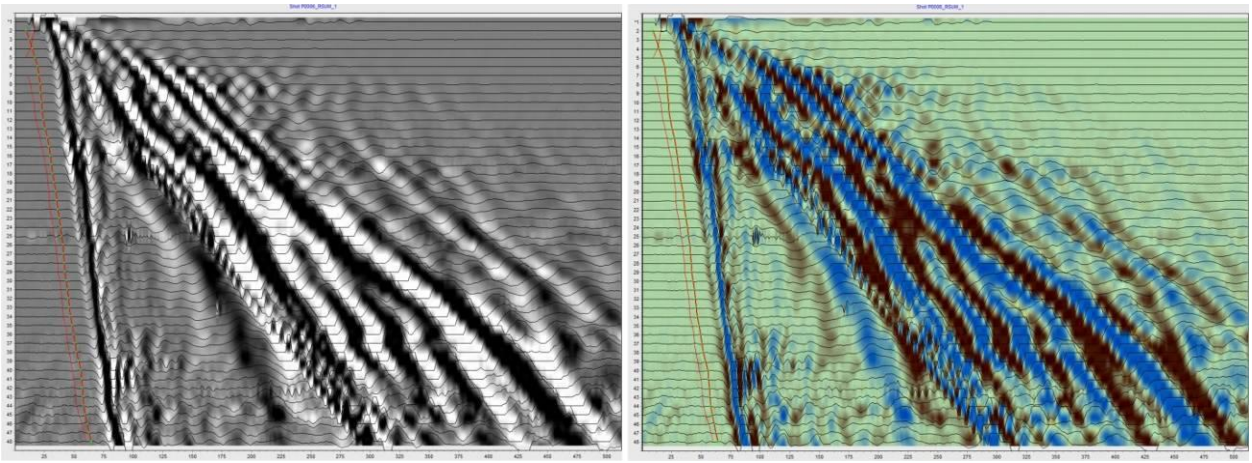
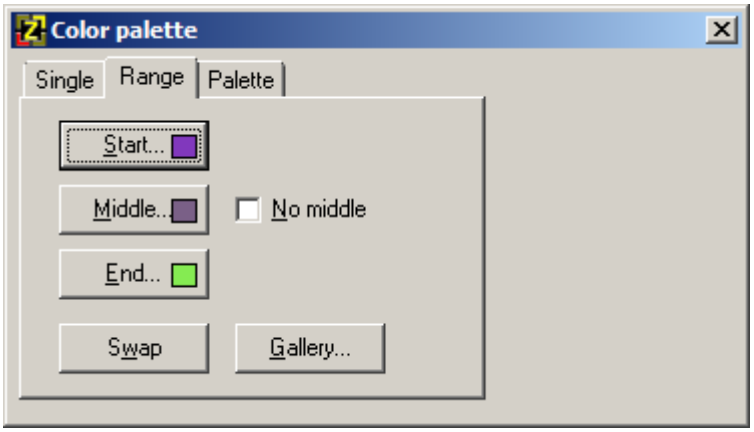


Fig. 13. Seismogram display options in an interpolated form.



*Pic. 14. Dialog **Color palette**.*

Seismogram picking mode main options

Seismograms can be sorted:

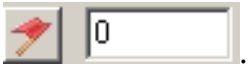
- by file names - **Options/Sort shots by filename**;

- by source positions - **Options/Sort shots by position**;

- by distance from a source - **Options/Sort trace by offset**.

Gain factor changing of the certain seismic trace is performed with a mouse wheel when hovering the cursor over it or in the **Table** tab of multitab section.

Seismograms *grading and summation* are performed when operating with the **Operations** tab of the multitab section.

To correct moment mark value, use the option *Start time* in the **Scaling options** window or in the **Hodograph** tab of multitab section using the field  . Options set the moment mark value (positive number in msec or counting) for all seismic traces. The axis of times/counting is displaced so that the moment mark corresponds an axis zero. Thereat appearance of negative values on an axis is possible.

The program provides work with separate seismogram traces when the mode **Active trace** is activated.

The first arrivals selection is performed in the **Picking mode** with the left mouse button. The right-click deactivates the chosen value on a hodograph. The program interpolates values of first arrivals between of first and second mouse clicking positions. The observed hodograph is displayed in red color, calculated (if you return to hodograph editing process after **Trace editor** inversion) is in blue.

The program allows displaying mutual first arrival times during seismogram picking time. To activate this function, use the following buttons:

To move to mutual traces of seismogram press the combination of [CTRL]+[ALT]+left-click on the necessary trace.

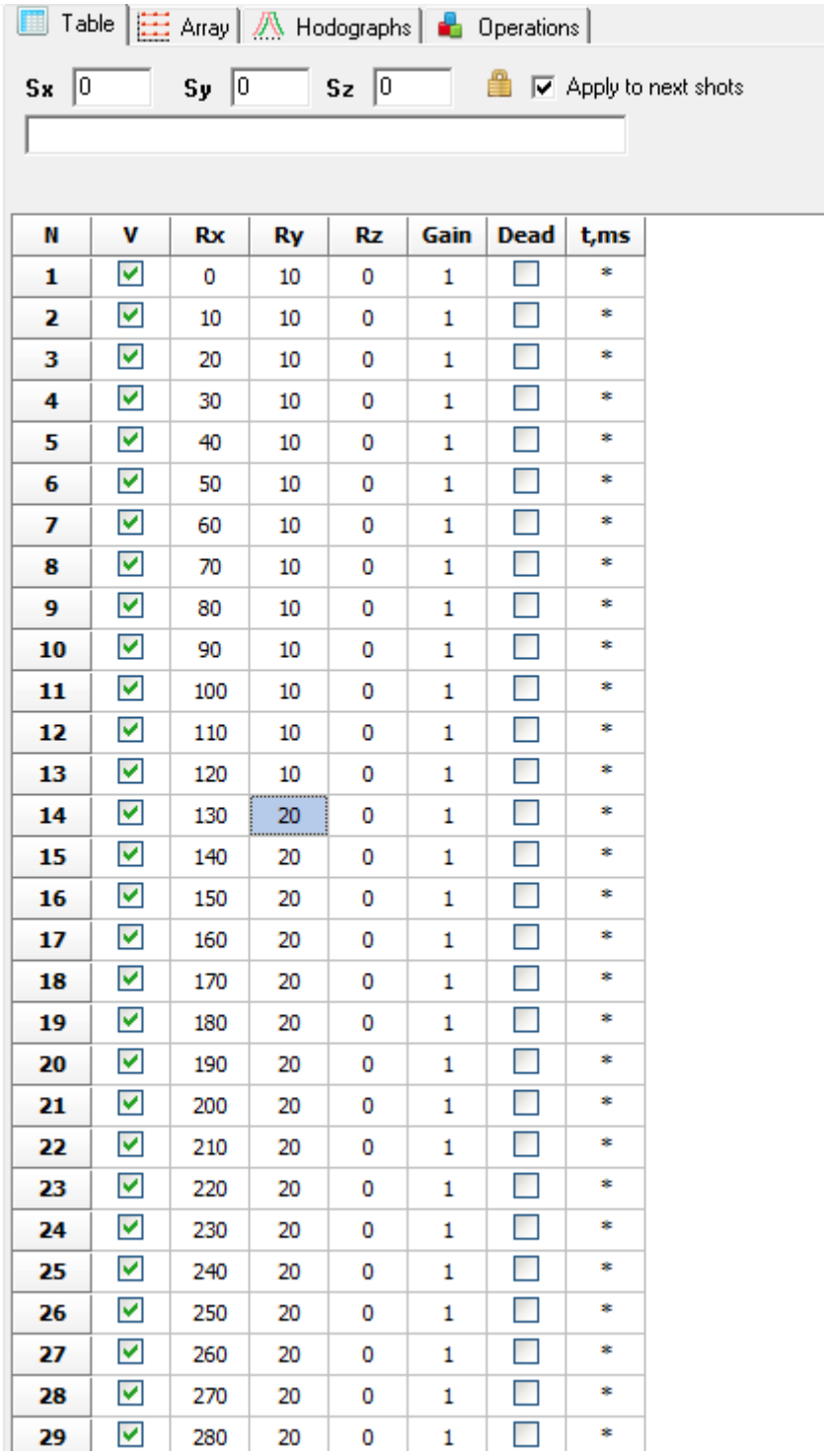
To cancel the action (picking, coordinates input), use the combination [CTRL]+[Z].

To create an inversion project and go to the inversion module, use the button on the toolbar. If you only want to refresh picking values in the project, use the button.

Multitab section options

The multitab section for survey configuration parameters setting and displaying contains the following parts:

The **Table** tab contains the table with the receivers coordinates, gain factors and the other settings of active seismogram traces (fig. 15).



N	V	Rx	Ry	Rz	Gain	Dead	t,ms
1	☑	0	10	0	1	☐	*
2	☑	10	10	0	1	☐	*
3	☑	20	10	0	1	☐	*
4	☑	30	10	0	1	☐	*
5	☑	40	10	0	1	☐	*
6	☑	50	10	0	1	☐	*
7	☑	60	10	0	1	☐	*
8	☑	70	10	0	1	☐	*
9	☑	80	10	0	1	☐	*
10	☑	90	10	0	1	☐	*
11	☑	100	10	0	1	☐	*
12	☑	110	10	0	1	☐	*
13	☑	120	10	0	1	☐	*
14	☑	130	20	0	1	☐	*
15	☑	140	20	0	1	☐	*
16	☑	150	20	0	1	☐	*
17	☑	160	20	0	1	☐	*
18	☑	170	20	0	1	☐	*
19	☑	180	20	0	1	☐	*
20	☑	190	20	0	1	☐	*
21	☑	200	20	0	1	☐	*
22	☑	210	20	0	1	☐	*
23	☑	220	20	0	1	☐	*
24	☑	230	20	0	1	☐	*
25	☑	240	20	0	1	☐	*
26	☑	250	20	0	1	☐	*
27	☑	260	20	0	1	☐	*
28	☑	270	20	0	1	☐	*
29	☑	280	20	0	1	☐	*

*Fig.15. **Table** tab.*

The options *Sx*, *Sy* and *Sz* set source horizontal and vertical coordinate (depending on the survey type the vertical coordinate can set topography (positive values) or depth (negative values)) for active seismogram.

The same coordinates are set in the **Set source positions** dialog.

Right-click allows setting coordinate increments for all the following seismograms.

The entry field below is intended for comments input for current seismogram.

The option *Apply to next shots* is intended to activate the general (through) trace parameters editing mode for the all seismograms. The general parameters are receiver coordinates, gain factors, indicator of channel unavailability.

The *V* column – activates/deactivates the seismic trace. Left-click on the column heading to activate, right-click – to deactivate.

The *Rx* column – sets horizontal source coordinate X. Right-click on table cell to set a coordinate increment for all the following cells.

The *Ry* column – sets horizontal source coordinate X. Right-click on table cell to set a coordinate increment for all the following cells.

The *Rz* column – sets receiver Z coordinate (in case of borehole observation – depth (negative values)). Right-click on table cell to set a coordinate increment for all following cells.

The *Gain* column – sets gain factor for seismic trace. When record polarity is wrong, you should enter gain with a minus sign. Left-click on column heading to increase, right-click - to reduce (modulo) all seismogram gain factors. Right-click on table cell to set gain value for all the following cells. In the *Apply to next shots* mode gains are set for all the following seismograms.

The *Dead* column – the channel failure indicator. This seismic trace won't be picked.

The *t,ms* column – sets the trace first arrival time value (in counting).

The **Array** tab contains the graph with the observation scheme (fig. 16).

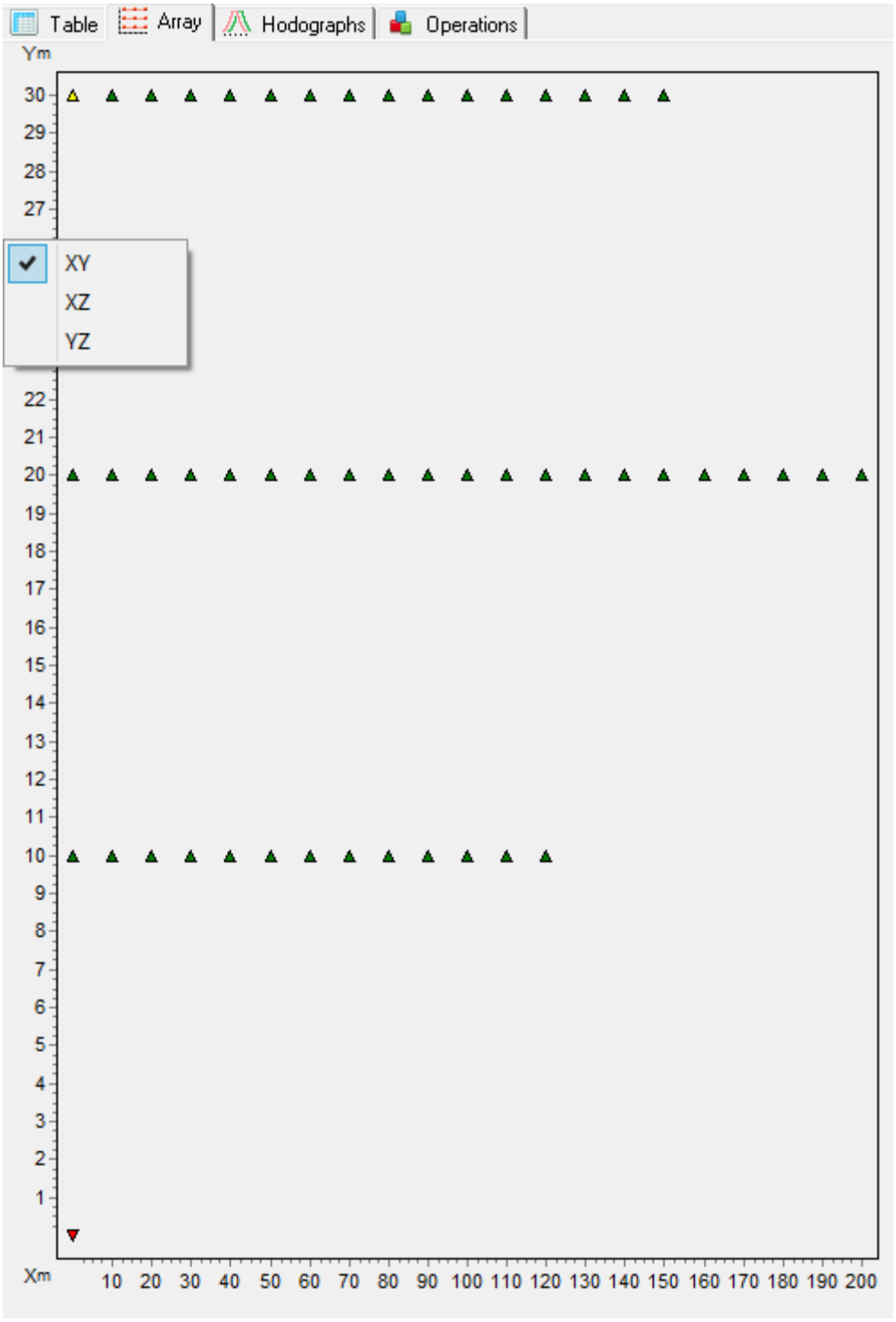


Fig.16. ***Array*** *tab.*

The shot point of active seismogram is displayed in red color. For the correct scheme displaying it is necessary to enter coordinates of receivers and sources. Accurate coordinates can be loaded from the file in the **Hodographs** tab.

[CTRL]+left-click the P-waves schematic position will perform transition to the corresponding seismogram of selected explosion point in the left seismogram displaying window.

The **Hodographs** tab is intended for all seismogram hodographs displaying and editing. During picking there are hodographs graphics in this tab. When pressing on graphic, the corresponding seismogram becomes selected (fig. 17).

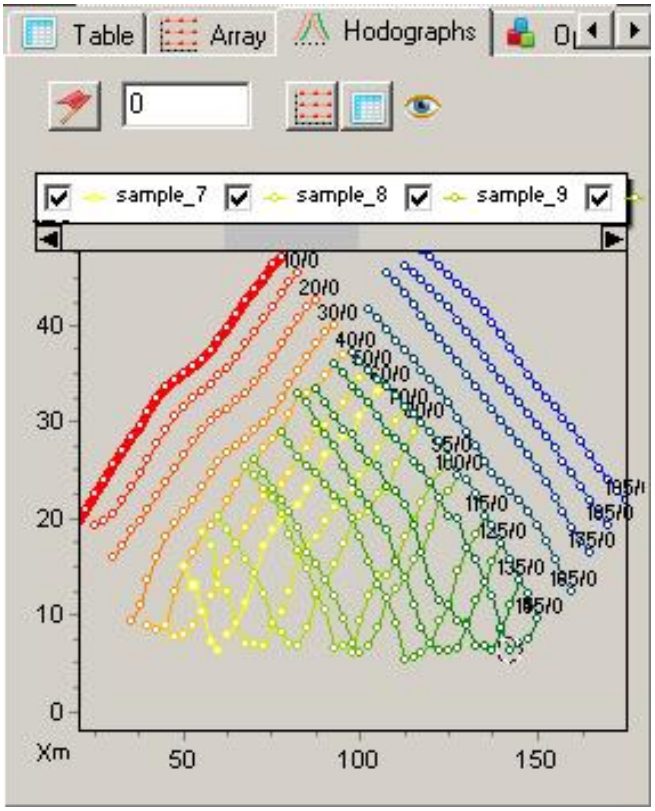
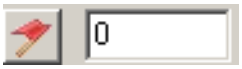


Fig.17. ***Hodographs*** *tab.*

Measured values editing is performed with moving graph point with pressed left mouse button. The whole hodograph is moved with the pressed SHIFT button.

To select one graph and remove the other, press the mouse button on a legend with pressed SHIFT button. By repeated pressing the inverse operation is performed. For graphs scrolling use the mouse wheel. To do it, select some adjust graphs (on a legend) and roll the mouse wheel having hovered the cursor over a legend. Indexes of active graphs will change.

The button  - sets value of time or the counting chosen by the slider in the seismogram window as a moment mark. The entry field allows setting a moment mark using keyboard. The times/counting axis is moved so that the moment mark corresponds to an axis zero. Thereat appearance of negative values on an axis is possible.

Buttons and also allow displaying the **Table** and **Array** tabs in separate windows, that significantly facilitates operating with data allowing seeing the full information about processed seismogram. In the **Array** window the same functions, as in the tab with the same name are performed.

The button activates/deactivates the whole hodograph. It is also possible to choose separate hodographs in the legend field.

The **Operations** tab is intended for displaying two seismograms (fig. 1). It is useful for comparison, summation seismograms, analysis of mutual seismic traces. The second seismogram is displayed by the line without filling and displayed according to scale of the first one. When displaying two seismograms simultaneously, it is necessary to set coordinates of receivers and sources.

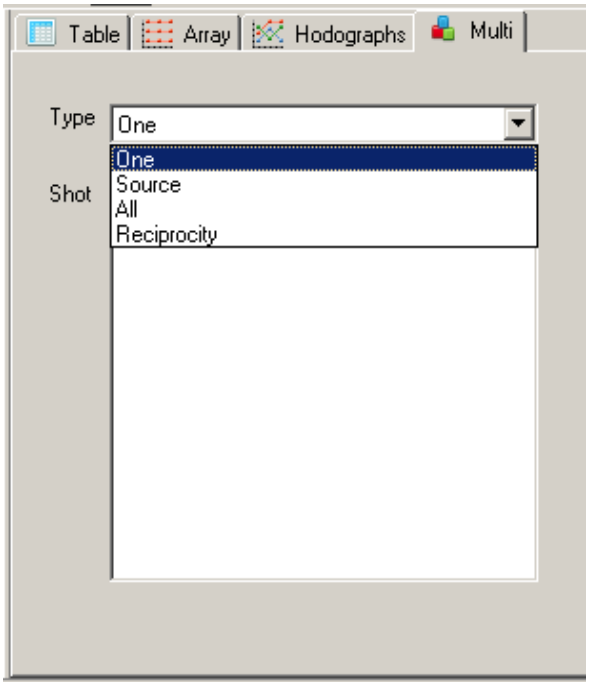


Fig.18. ***Operations*** *tab.*

The option *Type* sets displaying type for two seismograms. *One* value – display one seismogram. *Source* - one seismogram corresponding to the current position of a source (repetitions, accumulation) is used as the second seismogram. All the seismograms for an active source are displayed in the list field where you can choose them. *All* - any seismogram of the project is used as the second seismogram. All the project seismograms are displayed in the list field below where you can choose them. *Reciprocity* – display mutual seismogram (input of sources and receivers coordinates is obligatory).

Operations with seismograms

As a rule, the result of seismic survey is a file set of Seg-Y or Seg2 formats. These files are "raw" data to be prepared. After loading the data in the project it is necessary to input the survey geometry for each seismogram (if the geometry is not in the files). It can be simply done directly in the picking window (for a cable) table and using the option *Edit source pos* (for source positions) (Tools tab of the main menu).

Usually several seismograms exist (repetition, accumulation, left and right punches) at one source position. But only one is needed for interpretation which is obtained by summing the others, or selected from the best obtained at this point. When working with S - waves by the method of the left and right falls it is necessary to subtract from one another. To solve all these problems, use **Operations** tab. It allows you to visualize two seismograms and perform operations with them.

For operations with seismograms obtained at one source position, use *Source* option. In this mode, on the top of active seismogram another one is constructed, which is selected from the *Shot* list. Operation (summing/subtraction) is always held on the current and the one selected from the list. That is, to summarize, select the seismogram in the list, set the offset in the options if it is necessary and click the button . Use the button to remove unnecessary seismograms. In fact, after all necessary operations at the source point, you can remove all the seismograms but active.

The button is used to calculate the correlation coefficients between the active seismogram and the seismograms for the list. This essentially simplifies the selection of seismograms which will be summarize.

Option on the right right sets the seismograms offset (in counts) from the list relatively to active one which will be used during the operation. If you change this parameter, the second shift will move graphically.

Reciprocity mode allows you to display on top of the active seismogram another one which is built on mutual traces. This makes it possible to assess the quality of the field data, to identify errors in the recording start marks and control picking.

First arrivals picking

First arrivals picking is carried out in interactive and semi-automatic modes in the **ZondST3d**. Since it is impossible to automate fully first arrivals picking for the actual field data

of engineering seismic survey, the program has tool allowing simplifying and speedinh up the process.

Before start selecting the first arrivals, you should go to picking mode by pressing the button . Next step is the values choice directly on the seismogram which are displayed in a form of a hodograph. To add/correct a value use the left mouse button. To specify points sequence, use the left mouse button (click specifies the first and the last points, values among which are determined by linear interpolation). To remove a hodograph point, use the right mouse button. Depending on the handler preferences, the program provides a variety of ways to correct the times positions of first arrivals: by the front, maximums, minimums, and by the correlation maximum. Automatic picking mode is best used only for data with good quality. To activate first arrivals correction mode use the button . When right-clicking on the button pop-up menu of correction method selection appears. The following options are available: to null - reference to zero, to extremum - reference to the inflection point, to minimum - reference to the minimum point, to maximum - reference to the maximum point, Correlate - adjusts the first arrivals positions so that on the adjacent tracks points will be correlated with each other, Move to reciprocity point - values of mutual points are used.

In automatic correction the radius or the width of the width of search zone is important parameter. Search section is displayed as a circle around the cursor (when picking a single point) or a band – when picking a group values. Width/radius of the search area is edited by the mouse wheel while pressing [CTRL]. Selection of first arrivals is performed in the search area on the basis of the extremum type in semi-automatic picking. The correlation maximum mode, the program also determines the points on the adjacent traces on the basis of this criterion, and to the correlation loss. When group values picking in the search band, the circles will show the points where the first arrivals value will be after the closure.

An additional function **Hodographs/Autocorrect current** of the main menu is designed to automatically adjust the first arrival times on the basis of the correlation of adjacent traces to the already-picked hodograph.

When picking hodographs you should pay attention to the mutual time value (the blue points on the seismogram). They serve as a criterion of picking correctness. For dense survey layouts (with a large number of explosion points at the receivers positions) control of mutual points can reach 50-60 percent. This allows on the one hand correcting picking and automatically correcting errors in the determining of the start record marks (*Correct Start times* option) to another. If there is a large misfit between mutual points - you should find the reason which it is

associated with. To go quickly to the point of mutual seismogram use the right mouse button with the keys.

Thus, picking process is performed in the following way:

1. Pick individual seismograms interactively or automatically (depending on the data quality).
2. Adjust the value on the basis of mutual times and the whole hodograph group analysis. Control the picking error by the option *Reciprocity error*.
3. Adjust the level of recording starts automatically for all the seismograms (*Correct Start times*) or manually for each one.
4. Average mutual times.
5. Smooth hodographs if it is necessary.

Data filtering dialog

Dialog represents the tool for the analysis of field seismogram spectra and filtration. In the left window part there is a graph displaying the calculated spectrum of the working window and a filter form. To change the filter form, use the right mouse button (to the closest filter bend point).

The main aim of field data filtering is field signals selection against background noise. The program **ZondST3d** uses two filtering types based on Fourier and Wavelet transforms. The button runs the window allowing filtering of current seismogram. Seismogram spectrum is displayed in the top of **Fourier transform** tab. To calculate spectrum, use the button . If you select the option *All traces,* the average spectrum for the whole seismogram will be calculated, otherwise, spectrum will be calculated for the current trace.

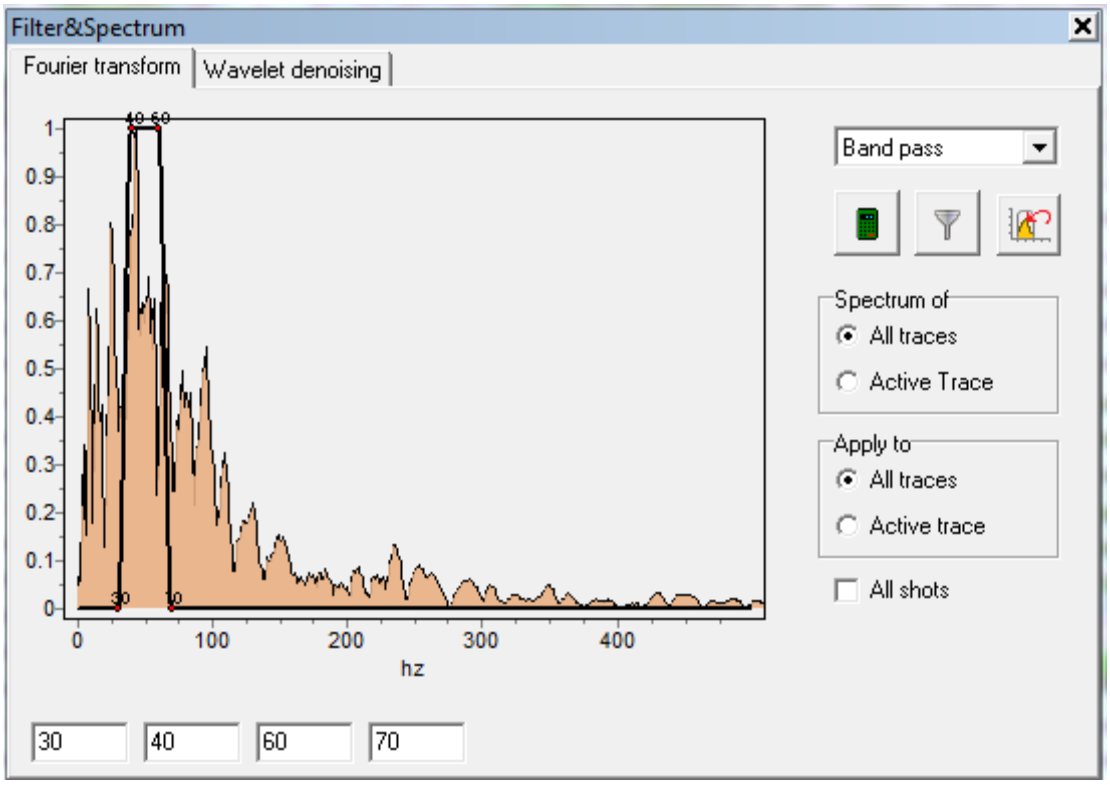


Fig. 19 Filter&Spectrum dialog. Fourier transform tab.

Then it is necessary to choose filter type (pop-up menu in top left window part). It can be a bandpass filter or low/high frequencies filter. To adjust filterб use the mouse (by dragging filter graph nodes (black line on top of graph line)) or in four entry fields below graphs.

Filtering can be performed after filter parameters selection (button). Filtering results are immediately displayed in the seismogram section, and also spectrum recalculating occurs. To return to the original seismogram use the button . Filtering with specified parameters can be applied to all the seismograms. To do it, select the option *All shots*.

To activate wavelet filtering algorithm go to **Wavelet denoising** tab. Options to control the wavelet filtering are the same as Fourier, with the exception of the following:
Wavelet type - specifies the wavelet type.
Wavelet order - specifies the wavelet order (complexity of its form).
Decomposition level - determines the number of small details that will remain in the signal after filtering. The higher the value, the larger signals will be filtered out.

The main stages of operating

Operating with new project algorithm can be divided into the following stages:

1. Survey type choosing (Survey type).
2. Source and receiver geometry setting.
3. Accumulation summarizing and removing unnecessary seismograms if needed.
4. Traces sorting in seismogram (Cable configuration).
5. Filtering and setting gain factors if needed
6. The first arrivals picking itself.

Field data inversion module

Data file creation and opening for inversion

To start interpretation in the **ZondST3d** program it is necessary to make data processing in the module of the first arrivals picking (**Create survey**) and to go to inversion mode , or create the data file with defined format, containing information about receiver coordinates, topography and first arrivals value.

Usually one file contains data of one observation profile. The text data files organized in **ZondST3d** format have «*.ST3» extension.

Zond data file	Open data file or project file with Zond format.
Program configuration	Open file with program parameters.

For the correct program work the data file shouldn't contain:

• the nonconventional symbols dividing records in string (use TAB or SPACE symbols)

• absurd values of measurement parameters (for example, negative first arrivals times values)

It is recommended, that the total number of measurements being contained in one file do not exceed 10000, and the number of unique sources/receivers positions do not exceed 500.

ZondST3D program can also assemble files of *.z2d format into one project. In order to do this in the dialog window «Open» in the «File name» field set *.z2d data types, press Enter and select necessary files. After that the **Multi-files case** dialog will become available (fig.20) and there it is necessary to specify types of the loadable data. The rest of modes collect data from several st3 files.

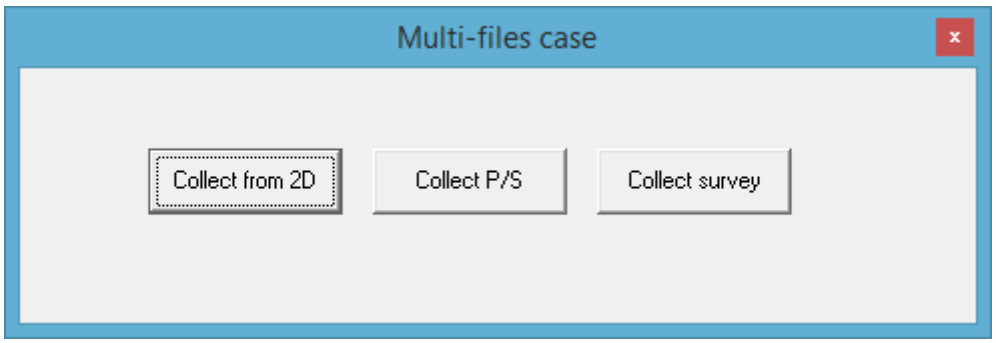


Fig. 20 Dialog of selection of loadable data types

Program allows to load the following data types:
Collect from 2D – assembles data of several profiles over the area. Usually 2D data has only X horizontal coordinate and after choosing this option for every profile, the coordinate setting dialog will become available (fig. 21). There it is necessary to set coordinates of the beginning and the end of every chosen profile.

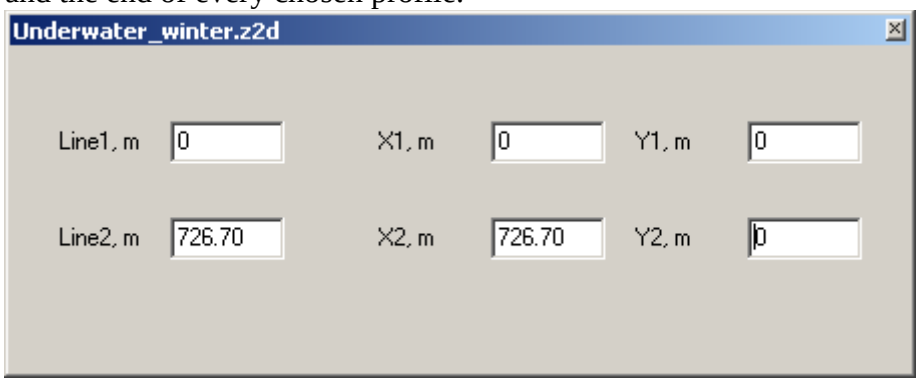


Fig. 21 Dialog of coordinates setting for 2D profiles.

Collect P/S – use this option if P and S picking data are in two different files.
Collect survey – the option makes it possible to merge data from different areas into one project. After selecting this option program will suggest to specify offset value in meters for every chosen data file.

When working with cross-borehole tomography data, vertical coordinate of electrodes should have negative value. For more information about the details of operating with this data type check the corresponding section.

To upload data from table text file or MS Excel file, pick **File/Import from text/excel**. After selecting file, you will see Export text data dialog (fig. 22). **Start** and **End** buttons in the top panel of the dialog window allow to set string of the beginning and the end of the data. Select the necessary string and press the corresponding button. Strings of the beginning and the end of the data will be highlighted in grey colour.

In the top panel there is also a drop-down list in which the data type should be chosen. Land – land survey data, Underwater – marine survey data, Cross-boreholes - cross-borehole data.

In the adjoining pop-up menu you should set the first arrivals units recorded in the file.

In the first raw of every data table column there is a pop-up list allowing setting the data type: line – number of measuring profile, Sx, Sy, Sz - sources, Rx, Ry, Rz – receivers coordinates, tp – first arrivals of P-waves, ts – first arrivals of S – waves.

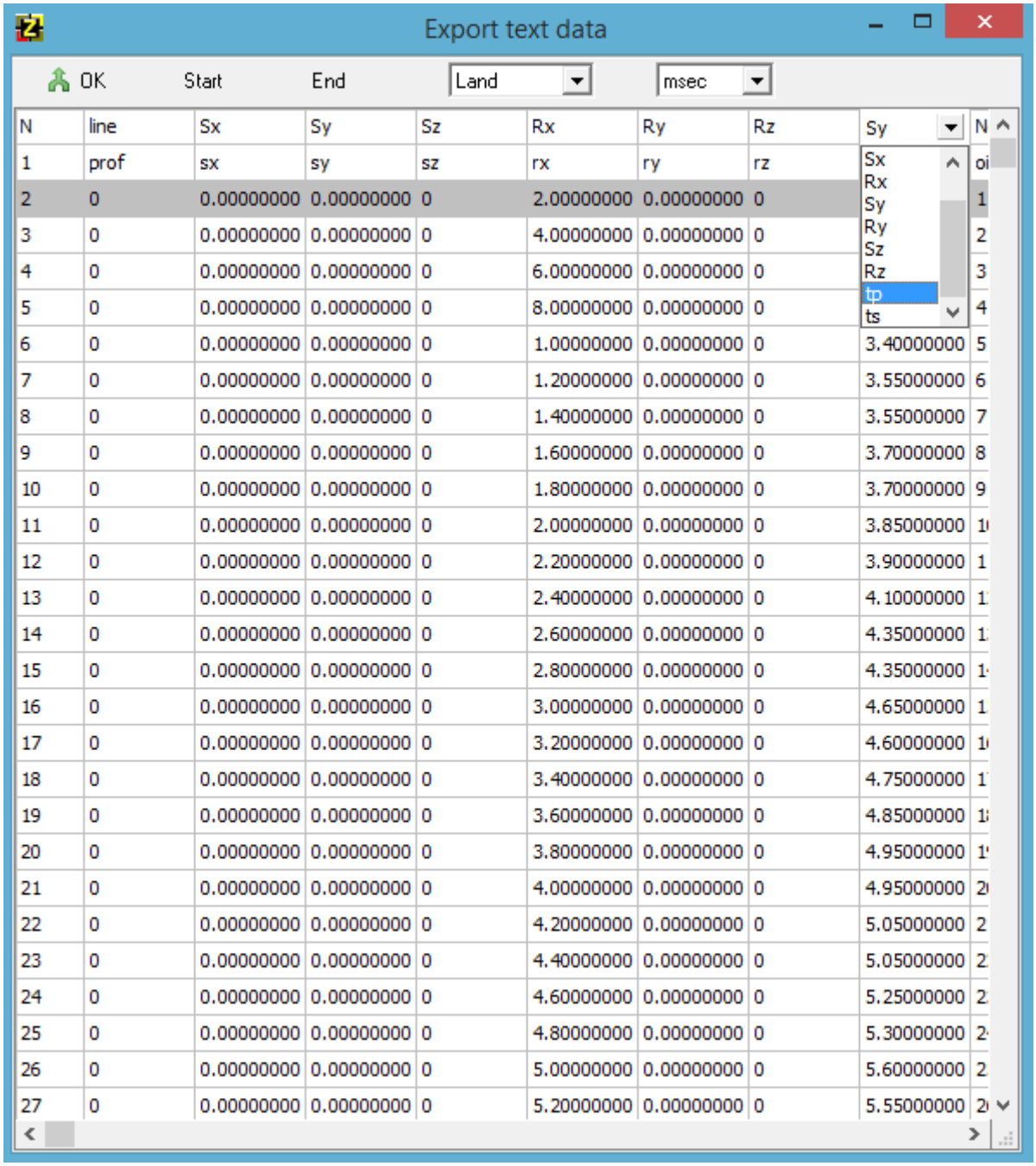


Fig. 22. Export text data dialog.

Press **OK** button and the data will be uploaded, and you will see a window of measurements parameters settings.

Setting and modifying tomography profiles and data

After uploading data file, setting profiles dialog **Set lines for survey** appears (fig. 23). This dialog lets you set, delete and modify profile lines (if 3D survey is assembled from 2D profiles), change coordinates of electrodes as well as upload picture or map as a base layer for the site plan. The dialog is available in the program main menu Options/ Set survey lines.

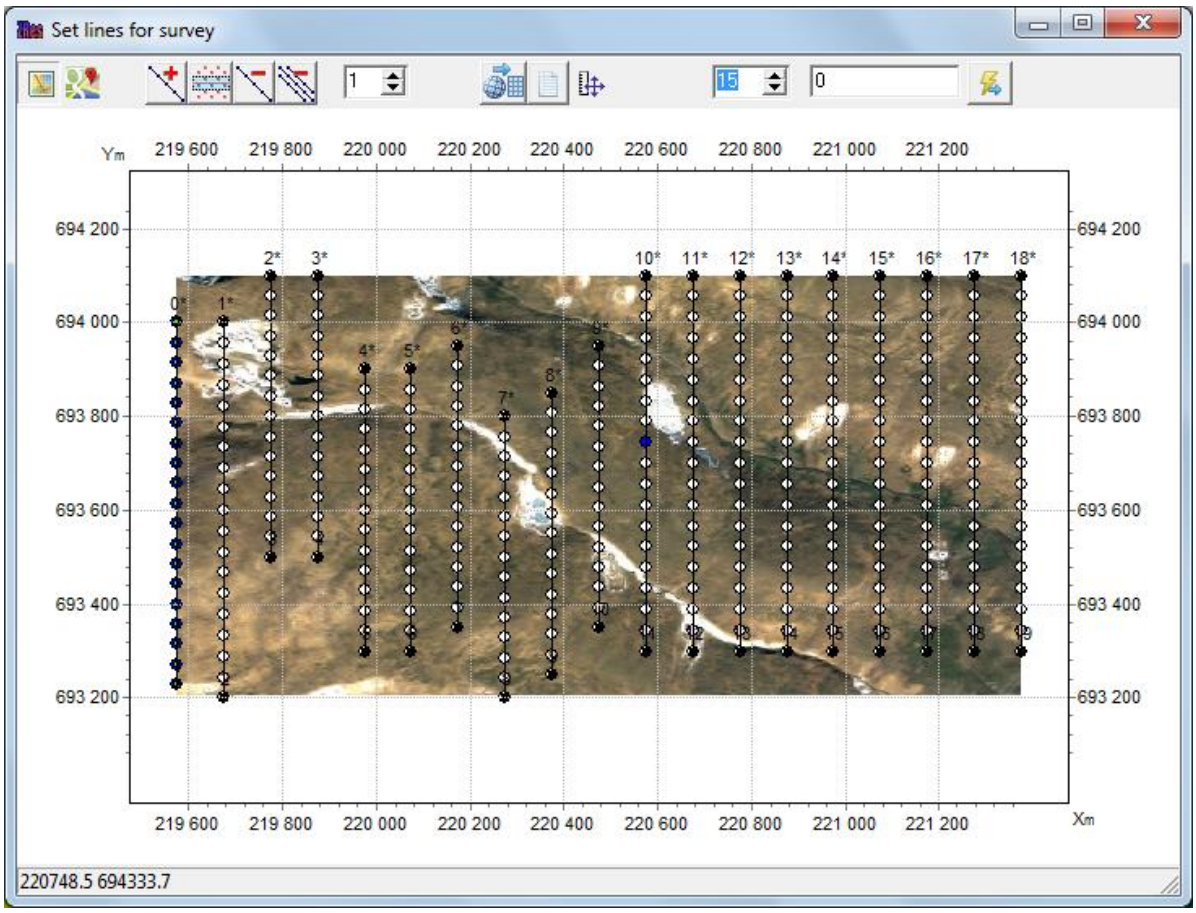
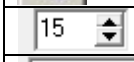
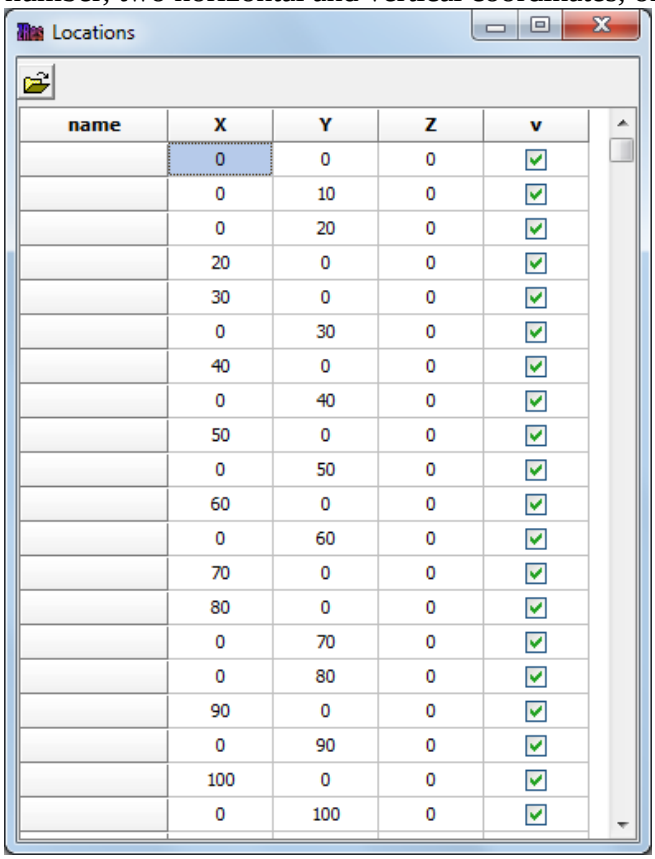


Fig. 23 Customization dialog for profile lines Set lines for survey

Панель инструментов данного диалогового окна содержит следующие опции:

	Map - load and display raster map file (fig. 24) **Topography -** display the relief isolines
	Load Google map
	Add profile line. To set profile lines click the left mouse button; to set the last point click the right mouse button
	Include measurement points in profile automatically. The points contained in a rectangular area around the specified line.
	Remove current profile
	Remove all profiles.
	Set the current profile number
	Convert from geographic coordinates to rectangle. If it is known a priori geographic coordinates (latitude and longitude) were in the file, it is necessary to convert them to rectangular using this button.
	Set coordinates in a table (**Locations** dialog)
	Select equal axes image scale or maximum window area filling.
	Set size of automatic selection measured points area in a profile
	Set the current profile name
	Go to the data inversion mode for selected profiles system

To change coordinates of receivers, use the button, which activates **Locations** dialog (fig. 24). Coordinates of electrodes can be uploaded from text file containing columns with site number, two horizontal and vertical coordinates, or they can be copied straight from any table file.



name	X	Y	Z	v
	0	0	0	☑
	0	10	0	☑
	0	20	0	☑
	20	0	0	☑
	30	0	0	☑
	0	30	0	☑
	40	0	0	☑
	0	40	0	☑
	50	0	0	☑
	0	50	0	☑
	60	0	0	☑
	0	60	0	☑
	70	0	0	☑
	80	0	0	☑
	0	70	0	☑
	0	80	0	☑
	90	0	0	☑
	0	90	0	☑
	100	0	0	☑
	0	100	0	☑

Fig. 24. Setting dialog Locations for coordinates of observation sites.

To finish operating with profiles editor and go to inversion, press the button.

Setting dialog for the start model

Customization dialog for the start model **Mesh constructor** appears after setting profile lines and pressing the button. This dialog is also available in program main menu **Options/Mesh constructor**.

The window includes two tabs - **XY view** and **XYZ view**.

XYZ view tab contains three-dimensional image of created mesh (fig. 25). To rotate the image, use the left mouse button. To scale the image, use scroll wheel. To move along the axes, use the right mouse button. Press Shift and hold it while right-clicking the axis to enable axes editor.

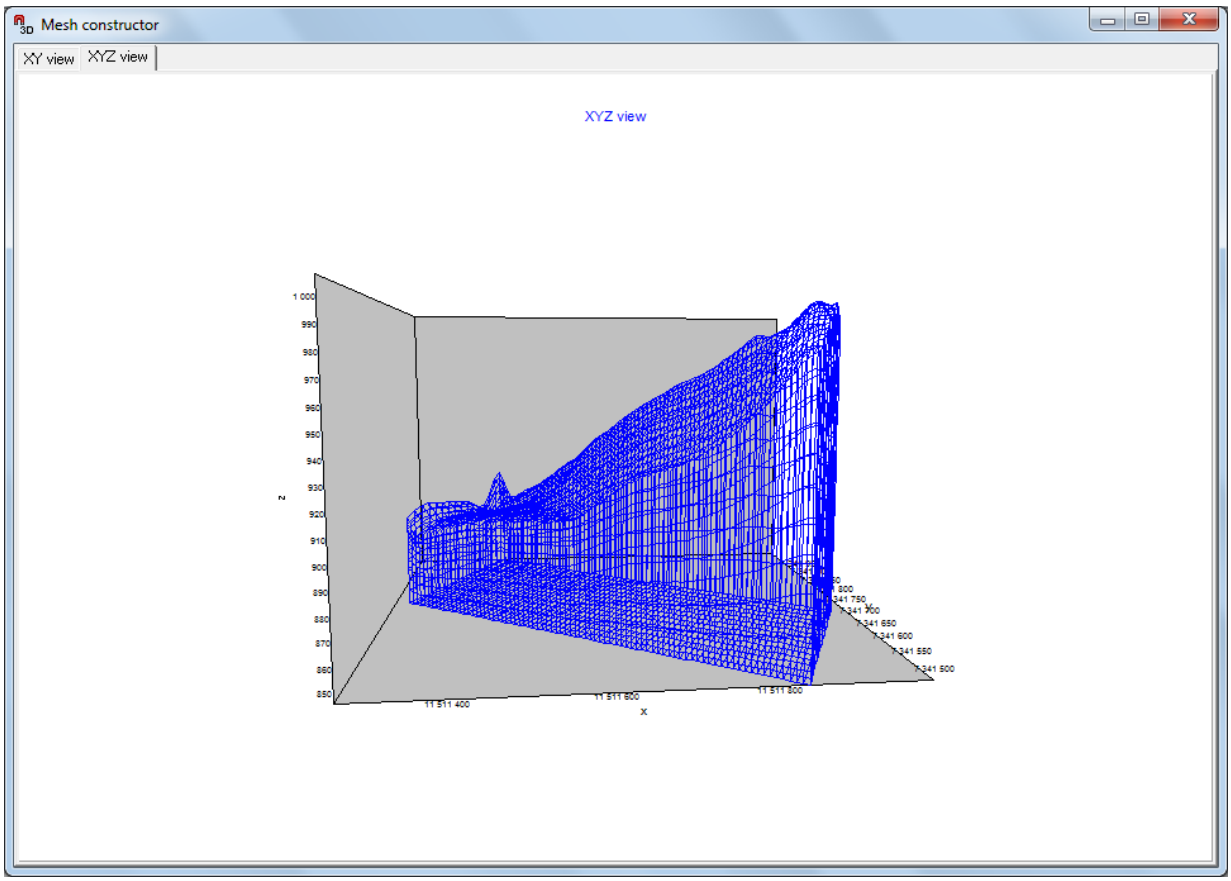
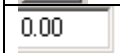


Fig. 25 Mesh constructor window, XYZ view tab.

In **XY view** tab setting of all mesh partition parameters is accomplished. The tab consists of two sections (fig. 26). In the left section there is a horizontal plan, the right section contains mesh partition settings. Toolbar contains the following buttons:

	Build a model in accordance with defined parameters
	Remove line. Hold the mouse pointer over needed line and click the left mouse button
	Add line. Hold the mouse pointer over the place where you want to create a line and click the left mouse button
	Show relief map as a base layer
	Rotate observation area by a defined angle
	Define rotation angle
	Undo rotation

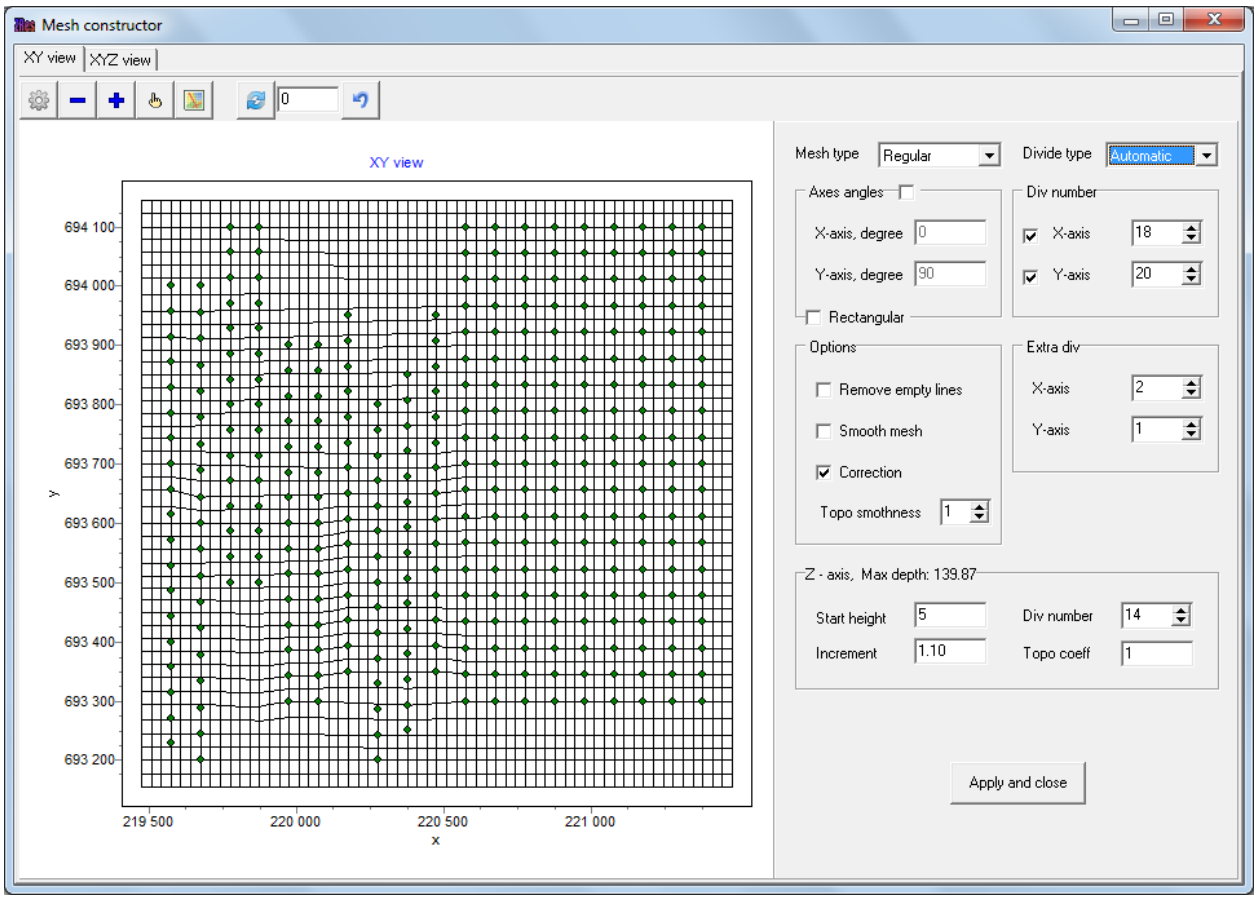


Fig. 26 Mesh constructor window, XY view tab.

Settings of mesh partition consist of:

Drop-down menu **Mesh type** sets mesh type. The following mesh types are available: *Regular* – fits for the majority of standard observing systems when electrodes are situated quite regularly, *General* – for relatively irregular observing systems, *One line* – in cases when observing system is considerably elongated in one direction (one or two long profiles). Program will add cells automatically.

Section **Axes angles** sets angles for X and Y axes (X-axis, degree and Y-axis, degree respectively). If Rectangular check box is selected, the value of rotation angle for second axis will change so that their mutual orthogonality remains.

Section **Div. number** lets you set a number of cells along X and Y axes.

Drop-down menu **Divide type** allows to choose automatic or manual mode for mesh setting.

If you choose **Divide type** – *Automatic*, the following sections are available:
Section **Options,** including the following check boxes: *Remove empty line* – remove empty cells (without measurement points), *Smooth mesh* - program will remove vertexes offset towards electrodes, *Correction* – program will find the optimal mesh increment if distance between profiles/electrodes is not constant, *Topo smoothness* – set the degree of relief smoothing (values from 1 to 5, where 5 – maximum smoothing degree).
Section **Extra div** allows to set a number of intermediate cells for X and Y axes respectively.

If you choose **Divide type** – *Manual*, section **Axes Range** becomes available, there you can set the minimum and the maximum coordinates values along X and Y axes respectively.

Section **Z-axis, Max.depth** is intended for setting partition along vertical axis. **Max. depth** – determines the depth of the lower layer. It should be borne in mind that the maximum depth value must not be too large because there is a decreasing with depth in the influence of magnetic and density section parameters. **Start height** – sets thickness of the first layer. This

value must comply with the required resolution. **Increment** – sets ratio of abutting layers thickness. This parameter is usually in the range of 1 to 2. **Div number** – sets a number of layers in the model. It is usually enough to have 12-14 layers to describe model. Setting too large values of this parameter is not desirable because it will lower computing speed. **Topo coeff –** set coefficient of topographic form distortion with the depth (0-5). 0 – relief of each next layer repeats the previous one. 1 – relief flattens out with the depth, the last layer is flat (fig. 27).

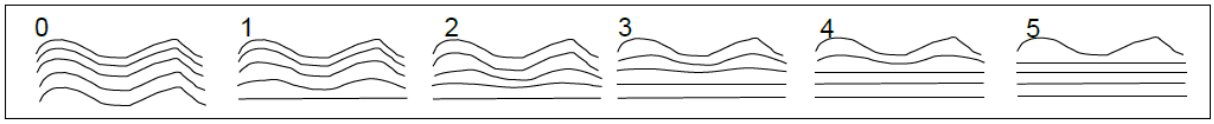


Fig. 27. Model layers distortion using Topo coefficient parameter from 1 to 5.

To finish operating with mesh constructor and go to inversion, press the

button.

Program main window

ZondST3D main window is divided into 4 sections (fig. 28). In the top left corner there is a 3D model section. In the top right corner observed and calculated graphs for selected profile are displayed (or, if the data is not structured, the general graph of S-R distance dependence). In the bottom left corner there is a plan of graphs or plan of contours for all the profiles. In the bottom right corner there is a 2D plan of model in the selected direction. Click the right mouse button in one or another section to see settings menu for this section. Settings are described in more details in corresponding parts of the guidance.

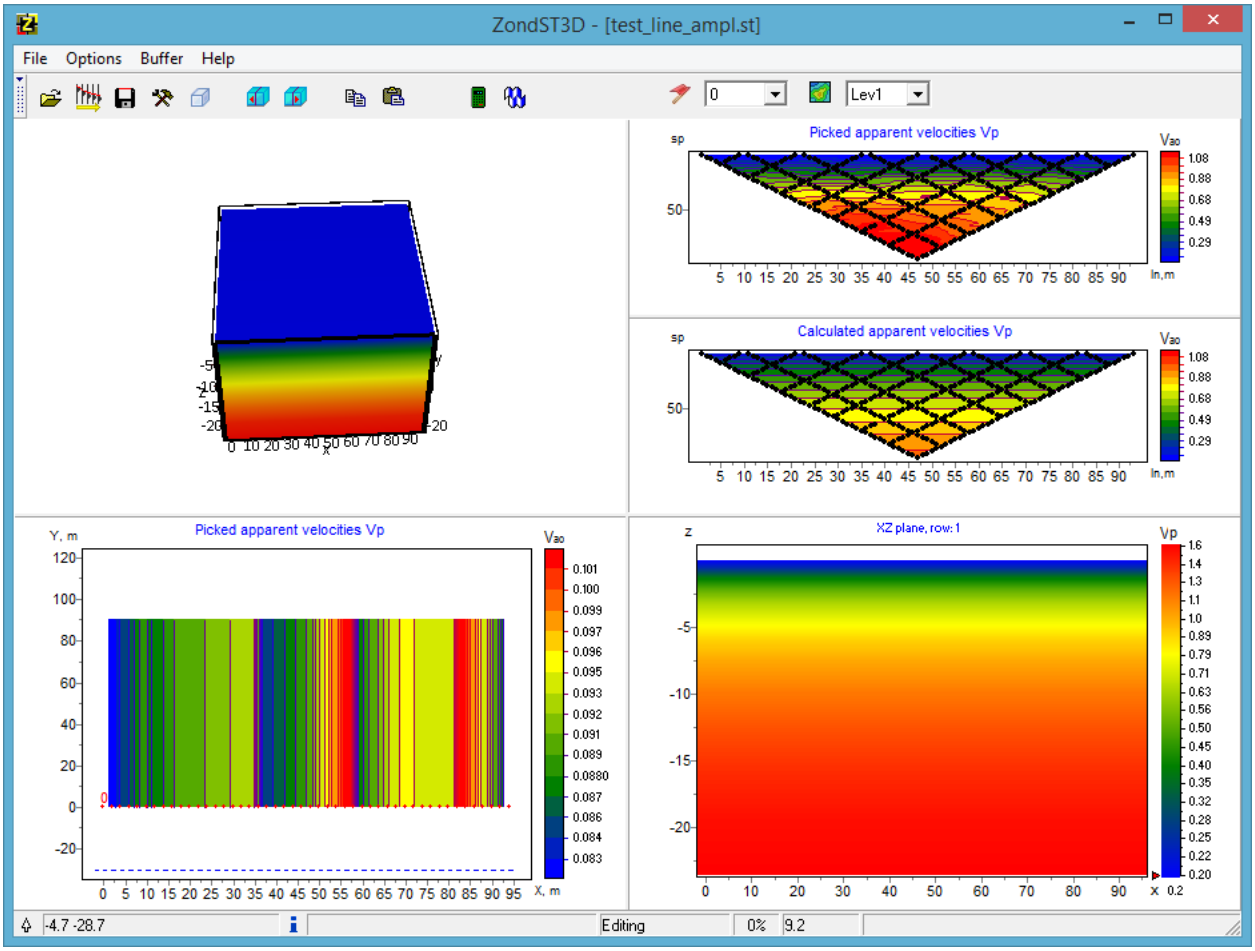


Fig.28 Main Window of ZondST3D

For all sections but 3D model zooming in or dragging some part is performed with pressed button (“rubber rectangular” tool). To zoom in a segment move mouse cursor down and to the right with left button pressed (fig. 29A). To return to primary zoom do the same but with mouse cursor moving up and to the left (fig. 29B).

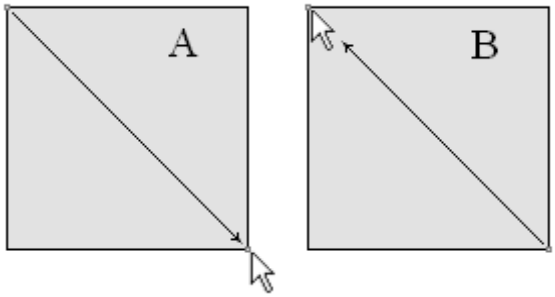


Fig. 29. Mouse cursor navigation to zoom in/out

The main window toolbar

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

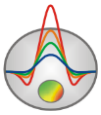
	Open data file.
	Save data.
	Run inversion parameters setup dialog.
	Run dialog for settings of 3D model image parameters.
	Go to previous slice of the model.
	Copy current slice of the model to clipboard.
	Paste data into current slice of the model. If right-click the button, there will be menu of options for filling the model: Current – paste model for selected parameter into current slice Current&+ – paste model for selected parameter into current and all next slices Current&- – paste model for selected parameter into current and all previous slices All params – apply selected paste regime to all parameters
	Run forward solution process.
	Start (one click) or cancel (second click) inversion process.
	Run data interpretation mode of resistivity method (DC).
	Run data interpretation mode of induced polarization method (IP).
	Two pop-up lists are intended to select current profile and data slice (corresponding to the same spans).

Functions of the main window menu

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

File/ Create survey	Open data file.
File/ Open file	Run synthetic measurement system generation dialog.

File/ Import from text/excel	Import a data file in text or Microsoft Excel format.
File/Save file	Save data.
File/Edit data	Open current data file in Notepad editor.
File/Print preview	Print main window of the program.
File/Reg file	Create register file.
File/Registration	Register program.
File/Recent	Recently used files.
File/Exit	Exit program.
Options/ Mesh constructor	Run starting model setup dialog.
Options/Set survey line	Call survey line setup window.
Options/Program setup	Run inversion parameters setup dialog.
Options/ Data/Pseudosection	Display observed and calculated data in pseudosection.
Options/Data/Graphics plot	Display observed and calculated data in graphics plan.
Options/Data/Observed data	Display observed pseudosection in second part of window.
Options/Data/Calculated data	Display calculated pseudosection in second part of window.
Options/Data/Data misfit	Display misfit pseudosection in second part of window.
Options/Data/Time arrivals	Display first arrivals values.
Options/Data/Apparent velocities	Display apparent velocities values.
Options/Data/Error gates	Display confidence interval.
Options/Data/With start times	Display with account of moments marks obtained during the inversion.
Options/Data/Show rays	Display rays in 3D model
Options/Model/Block section	Display model as blocks.
Options/Model/Smooth section	Display model in smooth interpolated graphic palette.
Options/Model/Plane	Choose model slice orientation.
Options/Model/Model histogram	Display model parameters distribution plot. Minimum and maximum values of parameter's colour scale can be specified in the dialog window.
Options/Model/Parameter	Velocity – display velocity model Anisotropy – display anisotropy model Attenuation – display medium attenuation parameter. When running the forward or inverse problem solving procedure in Attenuation mode, the procedure is applied to the attenuation parameter.
Options/Model/PR factor display	If there are VP and МЫ models, the Poisson ratio distribution can be displayed.
Options/Model/Cut by rays	Do not display the cells which rays do not go through.
Options/Inversion/Smoothness	Definition of a level, relative to which smoothness of a calculated model is determined. Available values: Median - from data average; Start model - from a starting model (useful when a priori information is available) Previous - from the previous iteration. Diagonal filter – include diagonal smoothing component (use not



	only adjoining cells but also diagonal cells).
Options/Inversion/Optimization	Damping factor – additional optimization of regularizer at each iteration. This regime allows to speed up the process algorithm convergence but several times increases time of each iteration. Since time of inversion for large models is estimated at hours, in some cases it is recommended to disable this regime. Dec factor – to speed up inversion at each iteration, damping factor is divided by set coefficient.
Options/Inversion/Invert start times	Activates the automatic retrieval of moment marks fок inversion. The option should be activated if you have issues with moment marks detection. It allows setting the maximum error for moment marks detection.
Options/Inversion/Invert anisotropy	Velocities inversion with anisotropy retrieval. It is recommended for cross-borehole measurements.
Options/Attenuation/Frequency	This item appears in menu in the attenuation mode. It sets the average operating source frequency.
Options/Attenuation/Max spacing	This item appears in menu in the attenuation mode It sets the maximum span R-S, amplitudes for measurements with large span will not count in the process of inversion.
Options/Import/Export/Background 2D	Load background – загрузить графический файл подложки для 2D плана Remove current – удалить текущую графическую подложку
Options/Import/Export/Background 3D	Load background – загрузить графический файл подложки для 3D модели Remove current – удалить текущую графическую подложку
Options/Import/Export/Import mod3D	Import current 3D model in Zond3D format .
Options/Import/Export/Export mod3D	Export current 3D model in Zond3D format .
Options/Import/Export/Import mod2D's	Import current 2D model in Zond2D format .
Options/Import/Export/Export mod2D	Export current 2D model in Zond2D format .
Options/Import/Export/Save rotations	Save 3D model as a set of images for different rotation angles.
Options/Import/Export/Save slides	Save 3D model as a set of images for different positions of slice plane.
Options/Import/Export/Export line to z2d	Export model for current profile in program of 2D inversion ZondRes2D format.
Options/Import/Export/Output settings	Exported picture setup.
Options/ Boreholes/Load borehole data	Open and display a file with borehole log data and stratigraphic columns
Options/ Boreholes/ Create/Edit borehole data	Create/edit borehole data.
Options/ Boreholes/ Remove borehole data	Remove borehole data.
Options/ Boreholes/ Set column's width	Set width of borehole columns.
Options/ Boreholes/ Edit	Edit positions of boreholes and corresponding lithology columns.

positions	
Options/Extra/Remove data with big misfit	Remove data with large misfit. After choosing this option, there will appear dialog where you can set the threshold value of the misfit.
Options/Extra/Get synthetic for big misfit	Use calculated data for points with large errors.
Options/Extra/Graph for cable pos	Display hodographs in correspondence with the order set in cable configuration.
Buffer/ Model 1, 2….	Save the current model to clipboard.
Buffer/ Open	Open saved to clipboard models and compare.
Help/ About	About the program.
Help/ Context	Run manual.
Help/ Check for updates	Check for updates.

"Hot" keys

Cursor pad /cursor in model editor	Change active cell of the model.
Delete / cursor in model editor	Clear active cell.
Insert / cursor in model editor	Insert current value to active cell.
F / cursor in model editor	Fix active cell value.
X / cursor in model editor	Use "magic wand" tool to select domain.
V / cursor in model editor	Delete selected.
Up/down / cursor in model editor	Change current value.

Status bar

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

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

Modeling

Modeling is an important process that precedes the field work. It allows choosing optimal system parameters to solve a geological problem. Using a priori information the interpreter can model different geological scenarios, planning geophysical work.

Seismic tomography forward problem solution

The program has three options for forward problem solving, i.e. seismic rays trajectories calculating in an arbitrary two-dimensional medium. Algorithm type is controlled by the option *Calculation scheme (Model style)* in the program settings window (Program setup).

In the first case (*Velocity const for cell*) medium is divided into cells with specified

constant velocity. This means that the ray goes through the cell with a constant velocity. This algorithm produces the least smooth ray coverage.

In the second case (*Velocity const for node*) medium is divided by node set with specified velocity, between which its value changes linearly. This method has a smoother ray coverage in comparison with the previous one. Some problem is the reverse transition from the velocity values in nodes to the mesh model (after inversion). This procedure is slightly smooths velocities and increases misfit.

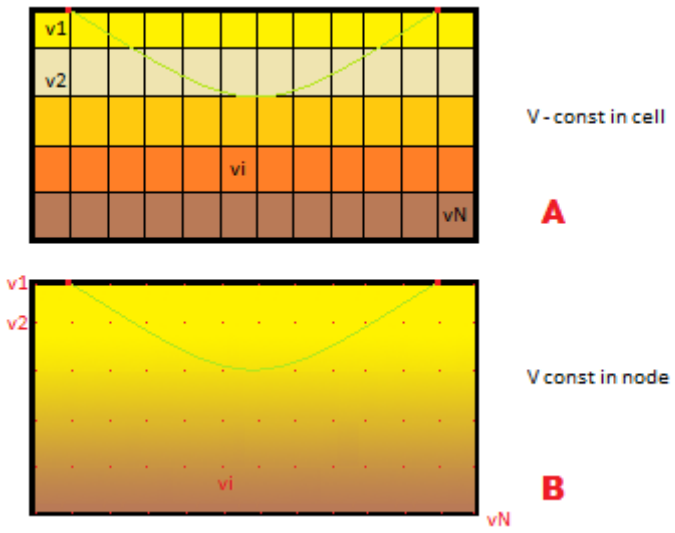
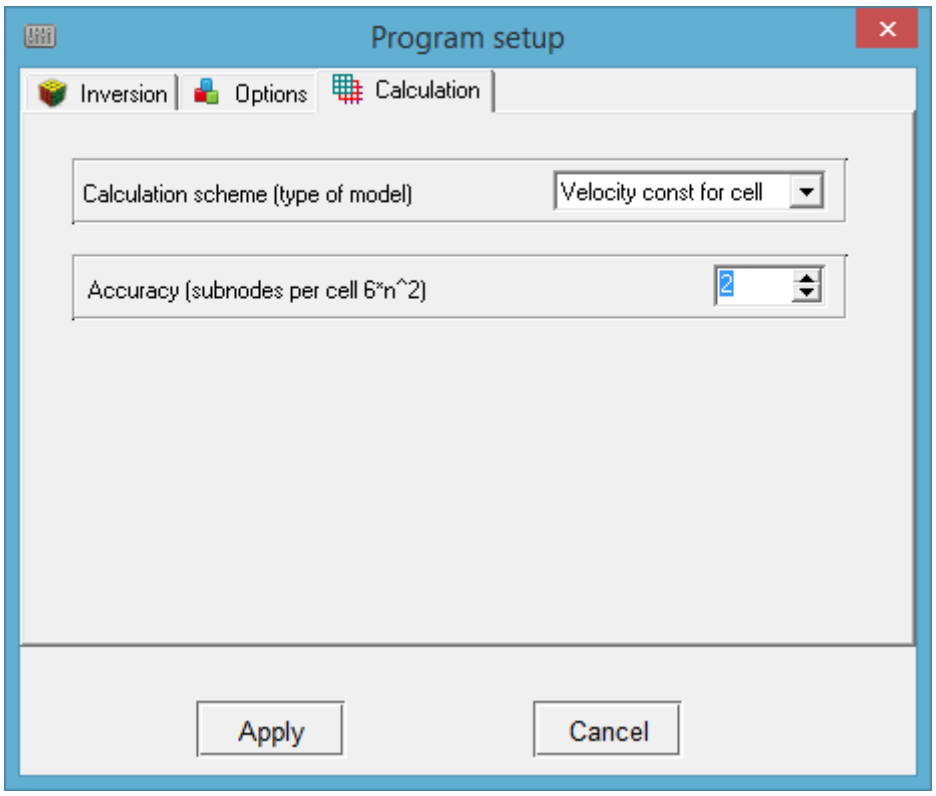


Fig. 30 Velocity specifying options.

Option *Accurancy (subnods per cell's side)* controls the accuracy of the calculations for all three methods. It sets the number of model intermediate calculating nodes, i.e. it determines the accuracy of the length dimension. Usually 3-5 nodes are enough for the precise solution. It should be noted that the calculation rate is directly related to the parameter. That is, the higher rate, the lower accuracy.



Synthetic survey configuration creation

The window is divided into three parts: the graph of source/receiver positions displaying (left), sources/receiver coordinates table (top right) and the current hodograph table (bottom right), which displays a list of source-receiver indexes pairs (fig.31).

The graph serves to display the unique position of sources/receivers and their indexes. You can choose (by the mouse) the sources and receivers. Active source is displayed in red, the current group receivers - in blue.

The coordinates table contain horizontal and vertical coordinates of the sources/receivers which can be edited. When right-clicking on the table cell, you can set coordinate increment for all the following points.

The current hodograph table contains indexes of source and group of receivers for active hodograph which can be edited. If you enter the wrong indexes, they are highlighted in red.

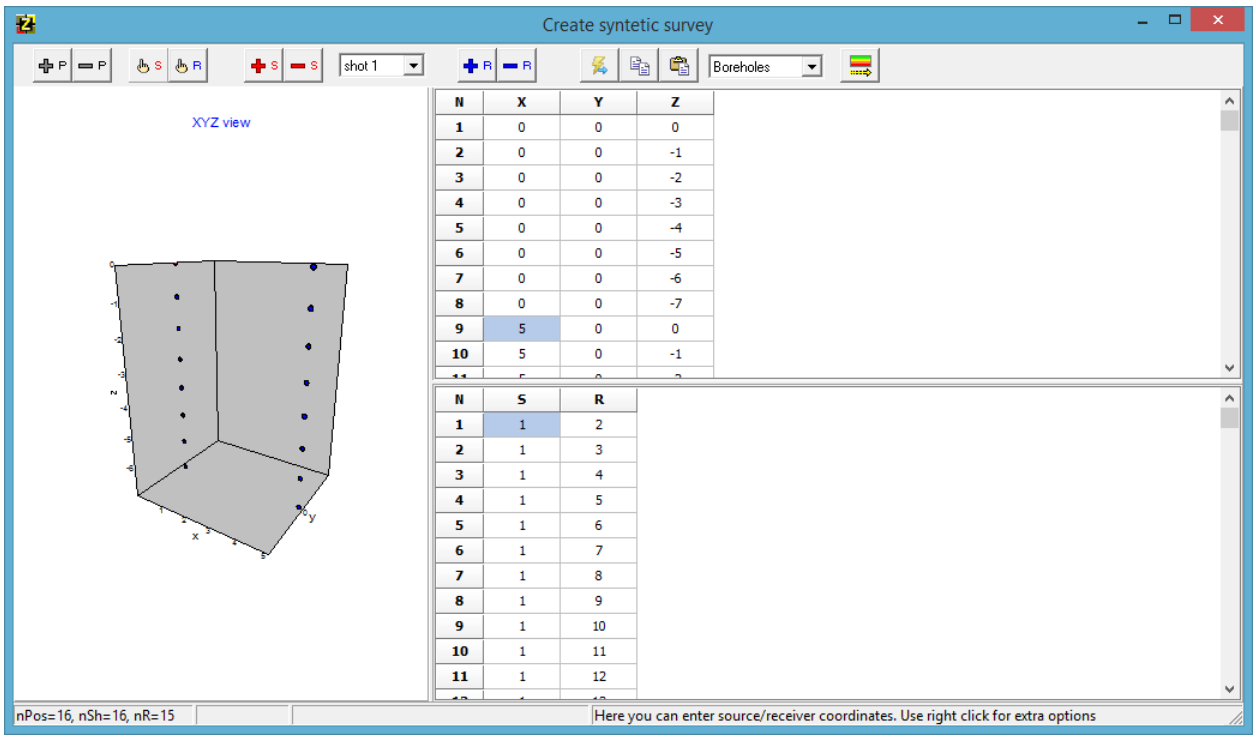
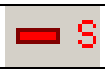
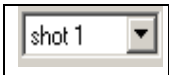


Fig.31. O Create synthetic survey program window.

The toolbar serves for quick access to the most commonly used functions. It contains the following functional buttons (left - right):

	Open file with sources/receivers coordinates.
	Save file with sources/receivers coordinates.
	Add new position of sources/receivers. New position is displayed in the window graph, coordinates – in the top right table.
	Delete the current sources/receivers position (in the table).
	Activate the current source selection mode. To select source position for the current hodograph, use the mouse in the left column. After selecting the source, the program switches to the receivers selection mode.
	Activate the current receiver selection mode. To select receiver for the current hodograph, use the mouse in the left column. If you select with pressed [SHIFT] button, new pair (source/receiver) is added to the button table. Otherwise, the current receiver will have new position.
	Add new hodograph. After adding, the program switches to receiver selection mode. To select the current hodograph, use the pop-up list on the right.
	Delete the current hodograph.

	Pop-up list for the current hodograph selection.
	Add a pair source/receiver to hodograph (in the bottom table).
	Delete the current hodograph pair source/receiver.
	Select receivers group positions which will be added to the current hodograph. To select use rubber rectangle in the right graph.
	Select receivers group positions, which will be replaced to the current hodograph. To select use rubber rectangle in the right graph.
	Call the pop-up menu functions of hodograph system automatic generation. *Select all positions* - Select all the positions, except for the source position to the hodograph. *Deselect all positions* - Remove all receivers from the hodograph. *Create full array* - Create layout survey, in which the source is located at each position and each source corresponds to the group of all the other receivers.
	Copy receivers indexes of the current hodograph to the buffer.

After creating synthetic survey configuration, press the button and grid parameters setup dialog appears. When you complete work with it, the function buttons for work with data activate, and summary information about data and model displays in the right-hand section of the status bar.

When you move the mouse cursor on the generated during the working windows coordinates corresponding to its own axis of the window displays the status in the status bar left section.

Main data file format

Program presents universal data format which consists of information about sources and receivers coordinates, relative relief elevation and first arrivals values and amplitudes itself.

Data file *ZONDST3D data files* (extention *.ST3).

For our purpose data file can be divided into 2 parts: 1) observed data, 2) topography data (if it exists).

I section of data file: observed data

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

The following designations (keys) for sources/receivers coordinates are accepted in ZONDST3D:

Sources **Sx Sy Sz**

Receivers Rx Ry Rz

Z sources/receivers coordinates can be specified if necessary. For example, in case of using buried source or marine measurements. When specifying sources/receivers Z coordinates it should be born in mind that negative values mean immersion below the measurement surface.

Positive coordinates are only used in case of marine measurements when source/receiver is on the surface or in a layer.

Profile index **Prof** where the measurement has been conducted (is used for separated data displaying along different 2D profiles).

For measured values:

Pft – first arrivals for P-waves (ms).
Sft – first arrivals for S-waves (ms).
amplp – amplitudes for P-waves (ms).
ampls – amplitudes for S-waves (ms).

Second and the following strings contain data corresponding to each measurement written in the same sequence as control keys in the first string.

Then (if necessary) horizontal mesh additional nodes column is filled. Each new node coordinate is entered after *** X Y Z symbol. Additional nodes are usually added for model extension beyond the area sources/receivers or if there is sharp relief beyond the area, it is also convenient to set additional nodes for cross-borehole tomography.

II section of data file: tomography data

If topography data is present, the next string contains word **topo** and is followed by a list of coordinates and relief excess values(X Y Z).

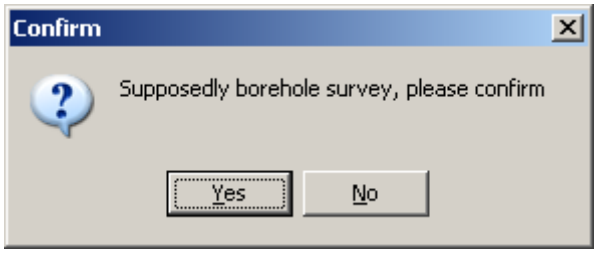
topow – this key is used if the interpretation is performed for marine survey data (on the water surface or at the bottom). In this case, bottom or bottom changing over to ground profile (if combined measurements are used). Thereat, in this string you should specify space-separated water level (relative to the bottom coordinates (in the topography coordinate system)), waves velocity in water, a number of additional divisions of the water layer (3-10). The last two parameters can be changed using the main program window menu. You can set combined systems when the measurements are carried out at the bottom and on the surface of water layer. To do so, you should set the vertical coordinates of sources/receivers in relation to bottom profile.

topo* – in water layer this key simplifies topography data input if the cable is on the water surface.

Features of operating with various data types in ZondST3D

Cross-borehole tomography data

For cross-borehole tomography data Z coordinate must be negative. When uploading file, program determines data type automatically and in case of borehole data you will see prompt to confirm:



In the resulting window **Mesh constructor** in the section of vertical Z-axis splitting the axis type should be set as **Borehole.** In this case program splits vertical axis automatically relying on a number of electrodes.

There is an example of 3D model for cross-borehole tomography project in the figure 32. To display all the electrodes, you have to check the box in 3D model settings **Show/Survey points**.

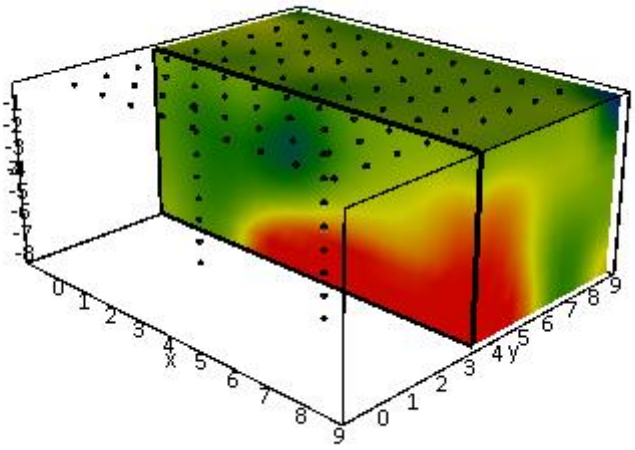


Fig. 32 An example of 3D model for cross-borehole tomography data.

Marine survey data

If you choose Underwater data type, there will be additional item Options/Inversion/Underwater options in the program main menu. It contains the following options:

При выборе типа данных **Underwater** в главном меню программы появится дополнительный пункт **Options/Inversion/Underwater options.** Содержит следующие опции:

Velocity	Specify velocity in water
Invert	It is desirable to have water resistivity values for marine survey data interpretation. Use this option if water resistivity data is absent.
Sublayers number	Specify number of water layer splits. It depends on water layer thickness.

Apparent parameters visualization

There is a possibility to visualize apparent parameters as pseudosection **Options/Data/Pseudo-section** and as graphics plan **Options/Data/Graphics-plot**. The current profile can be selected from the list on the task bar, the profile number corresponds to **Prof** index in the data file.

Mathematical algorithms are generally used for the first-time data rating because of large amount of data. Pseudosections and graphs plans can be used just to selectively estimate convergence of observed and theoretical parameters as a result of inversion.

Graphs plan

Graphics plan is used for apparent resistivity values visualization as graphs along profile.

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

Graph editor is run when right-clicking the mouse on the graph with pressed SHIFT. Right click on necessary axis with SHIFT button pressed to run axis editor. Right click on legend to the right of the graph with SHIFT button pressed to run legend editor.

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

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

Pseudosection

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

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

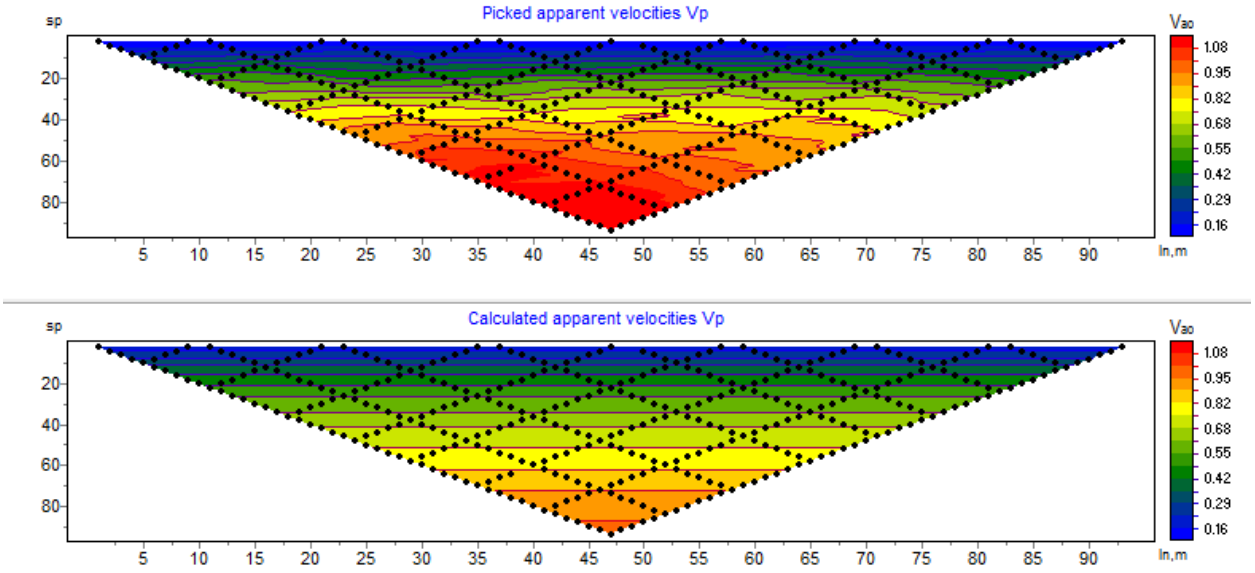


Fig. 33. Pseudosections of apparent resistivity: observed and calculated values.

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

Log data scale	Use logarithmic scale on color bar.
Smooth mode	Smooth pseudosection
Display grid point	Display measurement point ticks.
Display ColorBar	Display colour scale.
Setup	Run pseudosection parameters setup dialog.

Print preview	Print pseudosection.
Save picture	Save pseudosection in graphic file *. emf.
Save XYZ file	Save pseudosection in data-file.
X:Y=1:1	Make scales along the vertical and horizontal axes the same.
Default	Set pseudosection parameters on default.

Right click on necessary axis with SHIFT button pressed to run axis editor.

Zooming in or dragging some part is performed with pressed button ("rubber rectangular" tool).

Contour plan

The data slice is displayed in the program operating window under the 3D model. Observed parameter (**Options/Data/Observed**), theoretically calculated parameter (**Options/Data/Calculated data**) or misfits values for the area (**Options/Data/Misfit)** can be displayed (fig. 34). The slice depth can be changed using the list on the task bar.

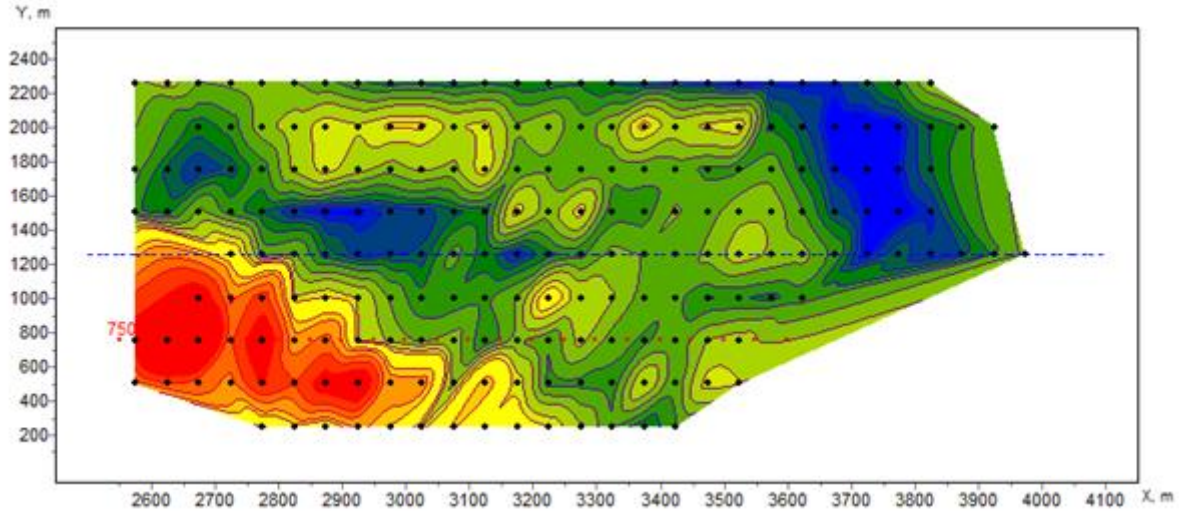


Fig. 34. Contour plan

Red line shows the position of the current profile for which there are graphs of observed and calculated parameter (top right corner section), blue dotted line shows the current position of model slice.

Right mouse button click on object section calls up the context menu with the following options:

Smooth mode	Use logarithmic scale on color bar.
Display grid point	Smooth pseudosection
Display ColorBar	Display colour scale.
Setup	Run parameters setup dialog (more).
Print preview	Print contour plan (more).
Save picture	Save contour plan to image file.
Save XYZ file	Save contour plan to Surfer format.
X:Y=1:1	Make scales along the vertical and horizontal axes the same.
Load background	Load image as a base layer (image file).
Load googlemap	Load Google map as a base layer.
Remove background	Remove current base layer.
Draw in Surfer	Export to Golden Software Surfer

Load background option allows to upload any bitmap file as a base layer whilst the picture becomes semitransparent. This option is very useful for comparing measured data with topographic maps, plots, geological maps and tectonic maps as well as with another methods data (geophysical, geochemical, etc.) which are performed in graphical format. There is an example of usage geological map as a base layer in the figure 35.

Load googlemap option allows to upload Google map in accordance with set points coordinates. After choosing this option the UTM zone window where you have to select zone number will appear (fig. 36).

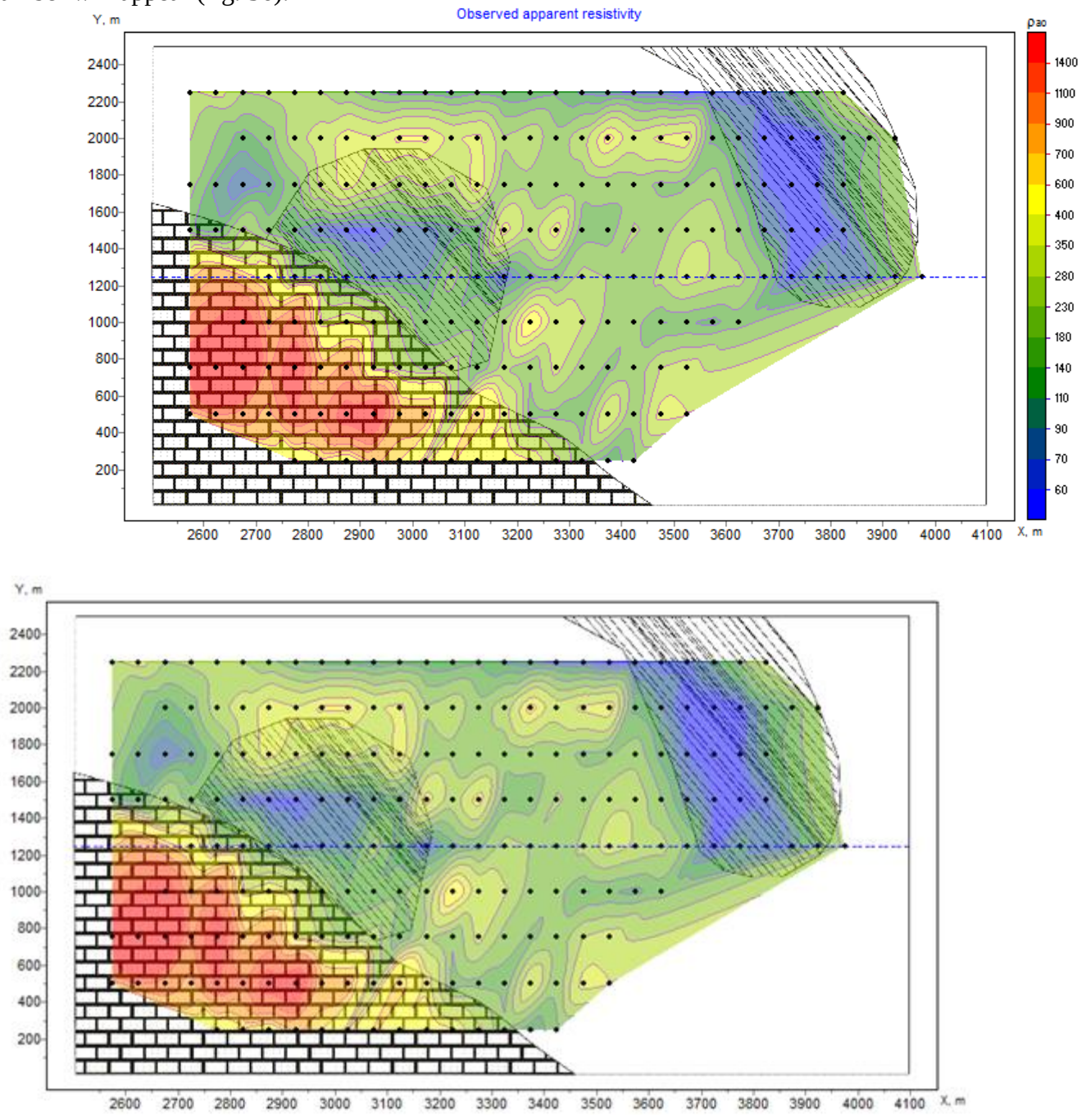


Fig. 35 An example of adjustment of the measured data contour plan and a geological map for the survey area.

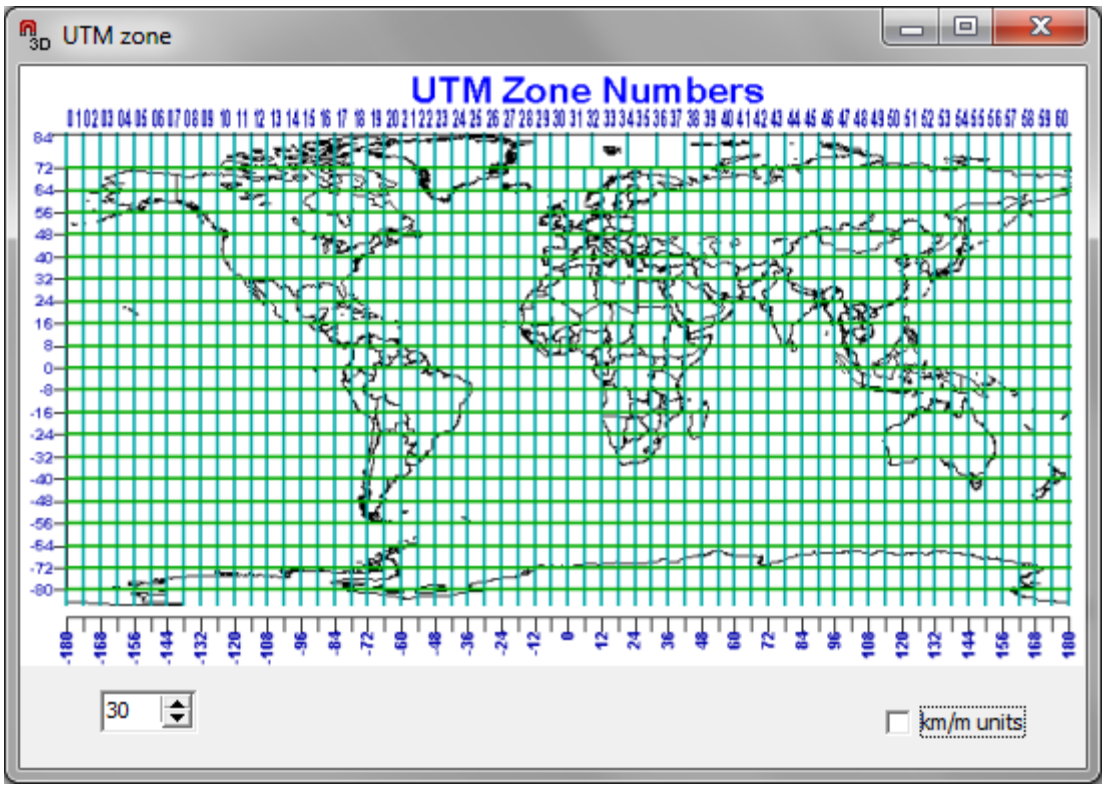


Fig. 36 Choosing UTM zone dialog.

Right click on necessary axis with Shift pressed and the axes editor will appear.

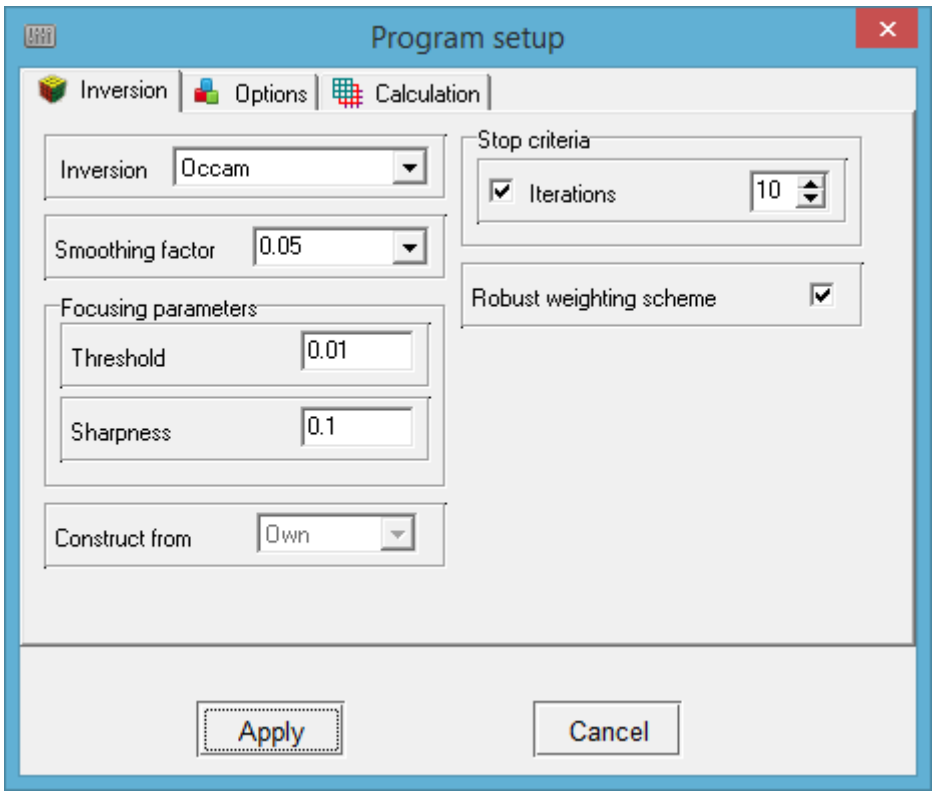
Data inversion

The next step after loading data file and start-up model setup is specifying inversion type and selecting parameters. Use button or **Option/Program setup** menu option to run inversion parameters setup dialog.

Inversion parameters setting dialog

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

Inversion tab serves for inversion parameters setup (fig.37).



Pic. 37. Program setup dialog window, Inversion tab.

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

Let us consider inversion algorithms by example of subsurface model that consists of several blocks (fig.38).

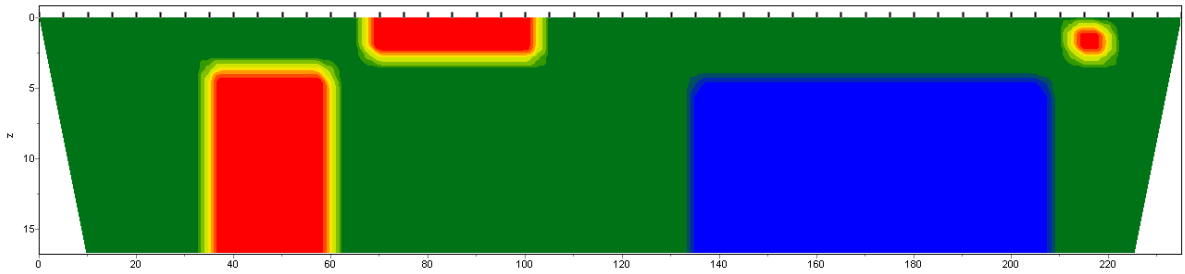


Fig. 38. Test subsurface model.

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

Smoothness constrained is inversion by least-square method with use of smoothing operator. As a result of this algorithm smooth (without sharp boundaries) and stable parameter distribution is received (fig.39).

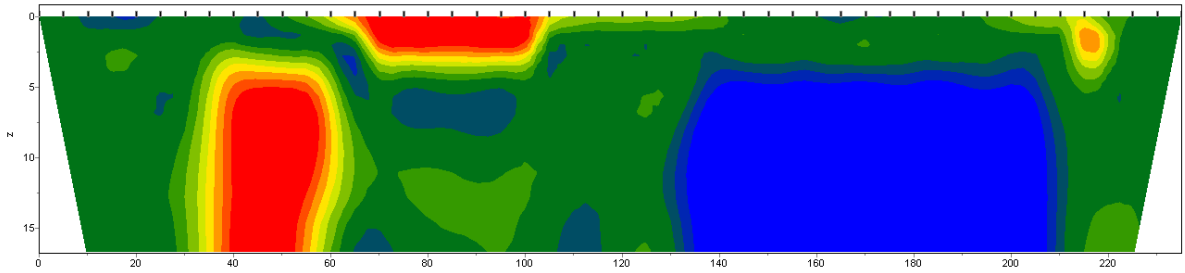


Fig. 39. Regenerated model as a result of Smoothness constrained inversion

Matrix equation for this kind of inversion is the following:

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

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

Occam is inversion by least-square method with use of smoothing operator and additional contrast minimization [Constable, 1987]. As a result of this algorithm the smoothest parameter distribution is received (fig.40).

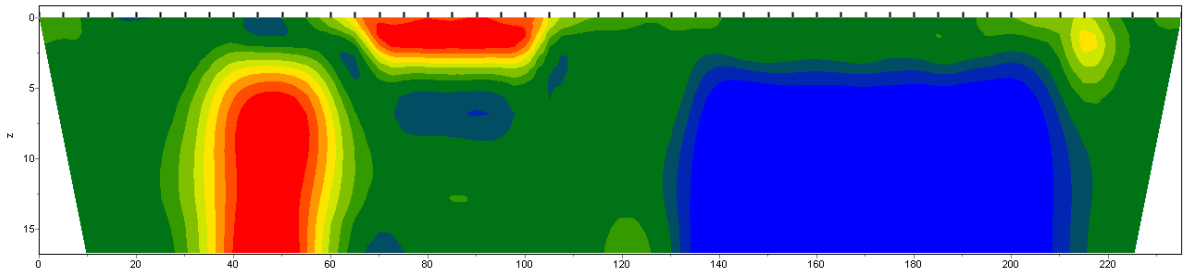


Fig. 40 Regenerated model as a result of Occam inversion.

Matrix equation for this kind of inversion is the following:

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

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

Marquardt – classic inversion algorithm by least-square method with regularization by damping parameter (**Ridge regression**) [Marquardt, 1963]. In case of little quantity of section parameters this algorithm allows receiving contrast subsurface model. (pic41).

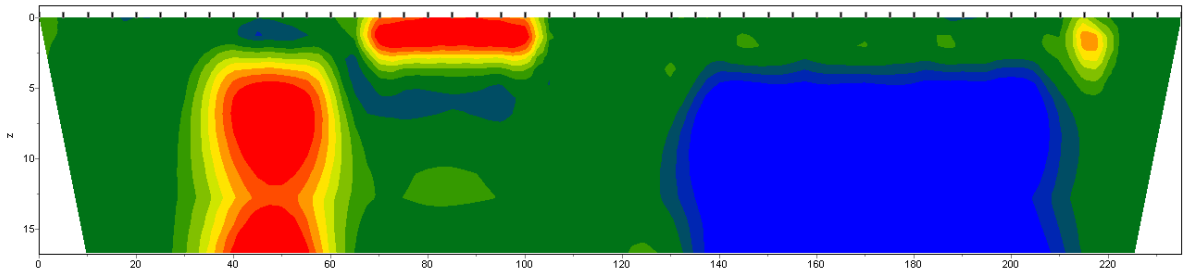


Fig. 41. Regenerated model as a result of Marquardt inversion

Matrix equation for this kind of inversion is the following:

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

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

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

Focused is inversion by least-square method with use of smoothing operator and additional contrast focusing [Portniaguine, 2000]. As a result of this algorithm piecewise smooth parameters distribution (that is model which consists of blocks with constant resistivity) can be received (fig.42).

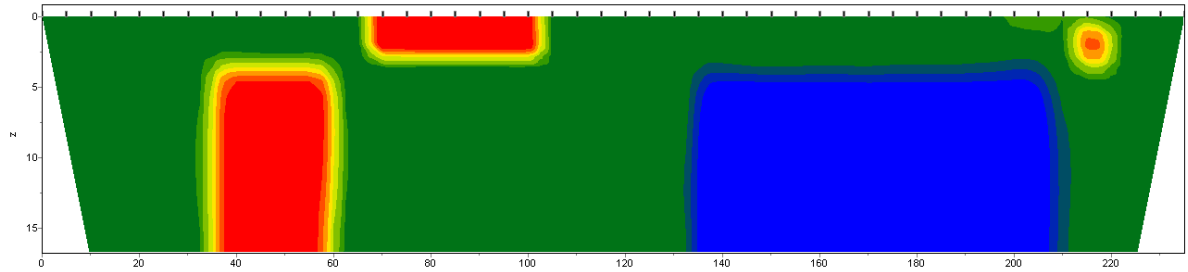


*Fig. 42. Regenerated model as a result of **Focused** inversion.*

Matrix equation for this kind of inversion is the following:

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

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

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

Blocks – fits parameters for certain domains which differ in resistivity. Domains with equal resistivity consider as single blocks (fig.43).

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

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

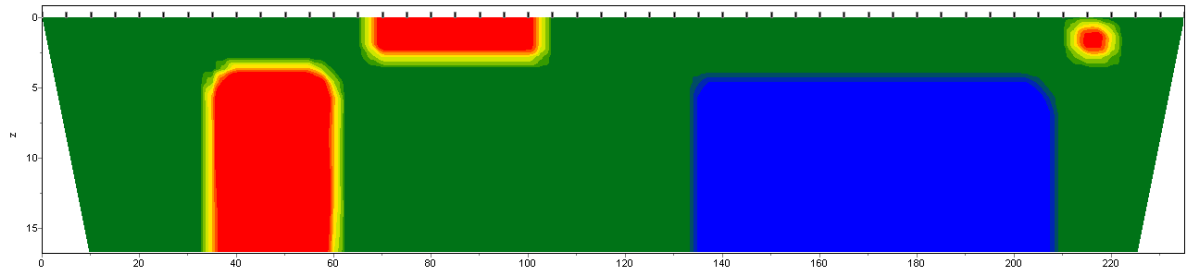


*Fig. 43. Regenerated model as a result of **Blocks** inversion.*

It is recommended to use this algorithm for more precise definition of already received results of preceding methods (***Focused*** is the best option) after merging cells in necessary blocks using **Cell summarization** function. Model editor should be used to select certain blocks manually: certain domains should be assigned with different parameters. Certain domains will be highlighted by boundary while working with this dialog window.

Smoothing factor sets dependence of measurement misfit minimization on model misfit. In case of noisy environment or in order to receive smoother and more stable parameters distribution quite high smoothing parameter value is chosen: 0.5 – 2.0; 0.005 – 0.1 values are used for high quality data. High smoothing parameter values mostly lead to high data misfits (fig. 44). Smoothing factor is used in ***Occam*** and ***Focused*** inversion algorithms.

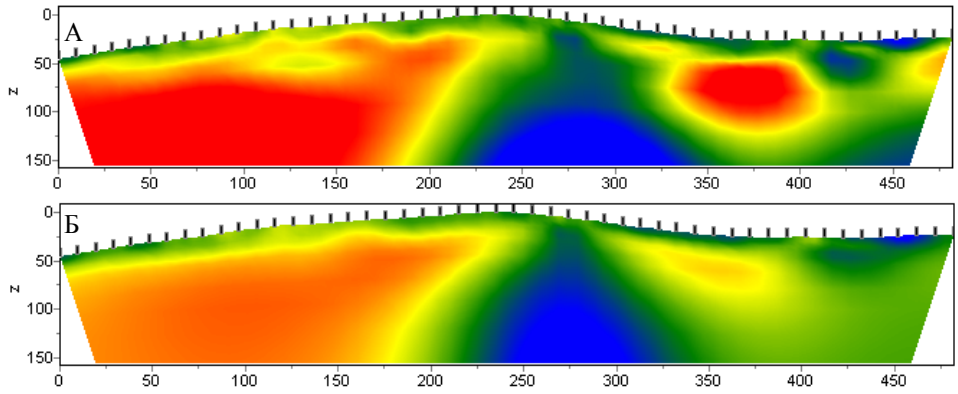


*Fig. 44. Geoelectric models as a result of **Occam** inversion with **smoothing factor**: 0.01 (A) and 1.0 (B). Resultant misfit for case A – 4.5 %, case B – 6 %.*

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

Stop criteria field contains inversion stopping criterion.

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

Focusing parameters field

Threshold – defines threshold contrast value for adjacent cells, and if it is reached adjacent cells parameters are not averaged (it is considered that there is a boundary between these cells). This parameter value is chosen empirically (0.001-1) (fig. 45). Small parameter value can cause algorithm discrepancy (in this case it needs increasing). Large value leads to receiving smooth distribution.

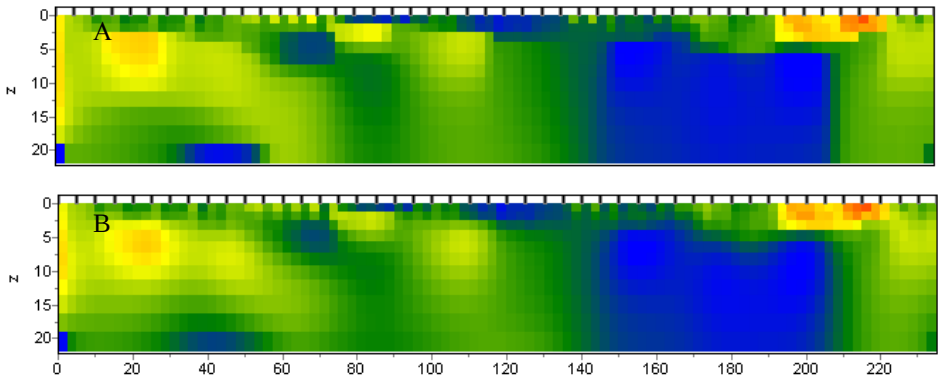


*Fig. 45. Geoelectric models as a result of **Focused** with **Threshold** parameter 0.01 (A) and 0.1 (B).*

Sharpness – defines ratio between minimization of causative body volume (0) and construction of piecewise-smooth distribution (1) (fig. 46). Value of this parameter is chosen empirically (0.7).

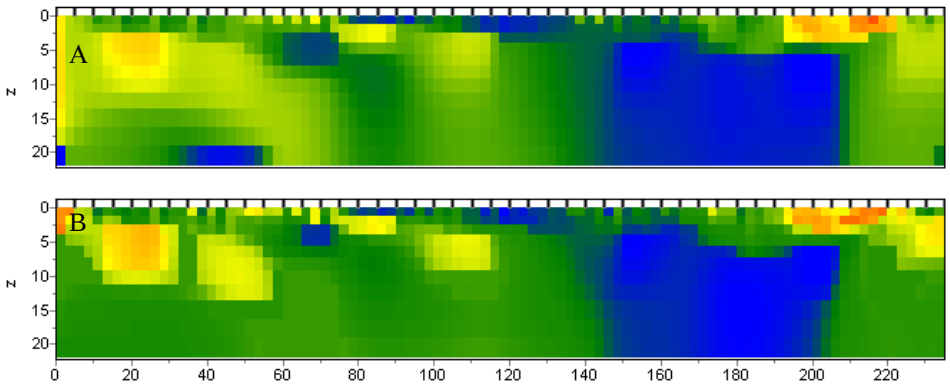


*Fig. 46. Geoelectric models as a result of **Focused** inversion with **Sharpness** parameter: 0.8 (A) and 0.2 (B)).*

Construct from – defines mechanism of focusing filter construction. If **Own** filed is selected, filter is constructed using current parameters (in current interpretation mode). If **Other** is selected, focusing filter based on other medium parameter is used.

Second tab **Options** serves for complementary inversion parameters setup (fig. 47).

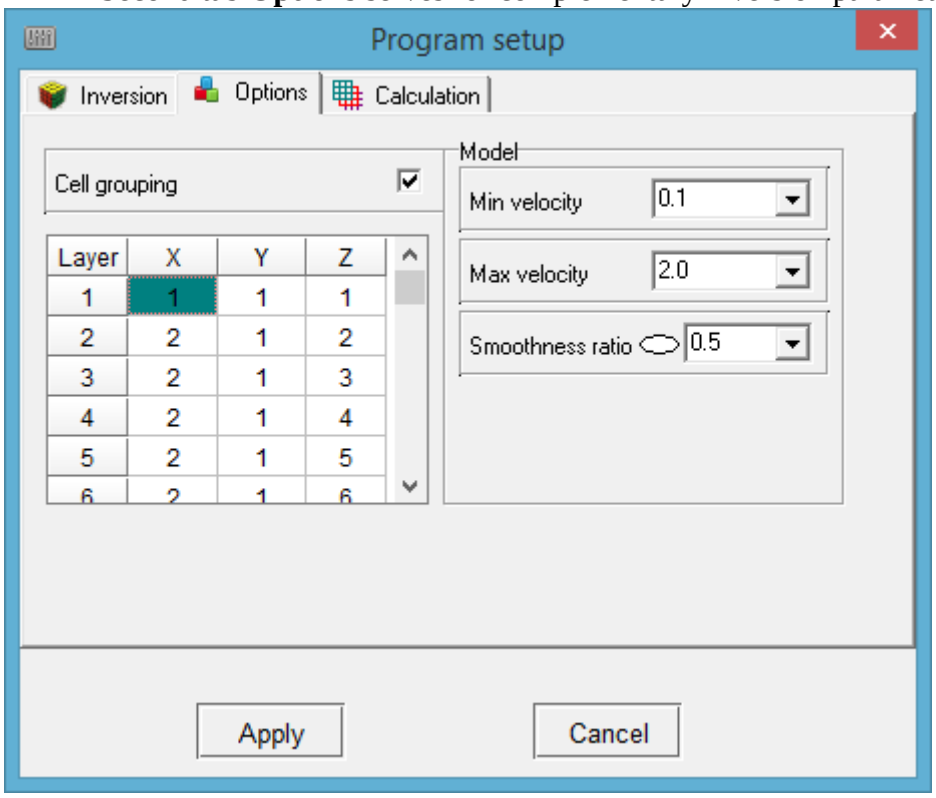


*Fig. 47. **Program setup** dialog window, **Options** tab.*

Область **Model**

Min …, Max … – устанавливает пределы изменения параметров модели при инверсии.

Smoothness ratio – specifies smoothness ratio in horizontal and vertical direction. Set this parameter larger than 1 for vertically-layered subsurface and smaller than 1 for horizontally-layered one. Usually this parameter value ranges from 0.2 to 1 (fig. 48).

Smoothness ratio – определяет соотношение степени сглаживания в горизонтальном и вертикальном направлениях. Для горизонтально-слоистых структур используйте значения этого параметра <1, для вертикально-слоистых >1. Обычно, для этого параметра, используются значения от 0.2 до 1 (рис.48).

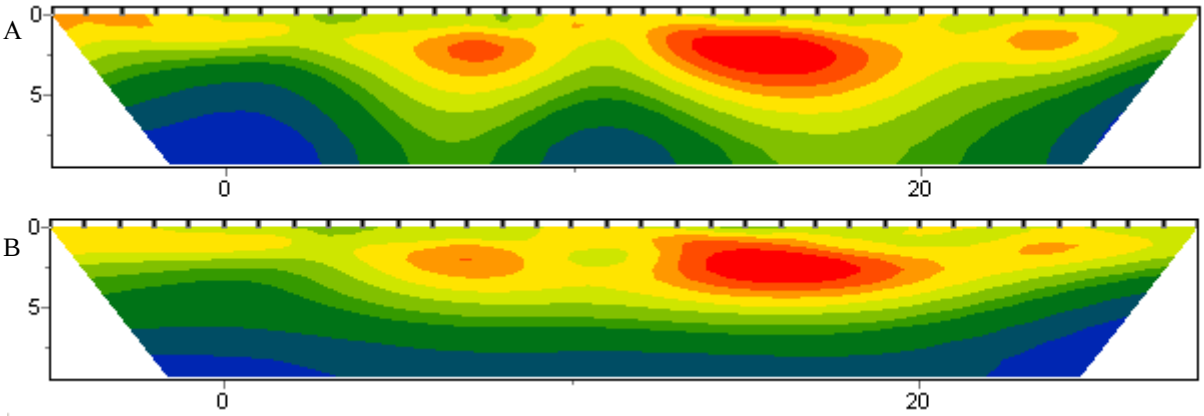


Fig. 48. Geoelectric model as a result of "smooth" inversion with Smoothness ratio parameter: 1 (A) and 0.3 (B).

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

The table contains four columns: in the first (**Layer**) there is a layer number for the initial model; in the following three columns (**X, Y, Z)** you can set a number of cells (in the corresponding direction) which are contained in every cell of the inversion mesh for this layer. The inversion mesh will be displayed in the model editor, while it is being customized. Double click on the cells in the X and Y columns allows to unite cells in the horizontal direction for this and all the underlying layers.

Matching mode anisotropy

Nowadays the simplest variant of seismic velocities anisotropy correction is realized in the program. Various values of section element velocities in horizontal and vertical directions are used for calculations. Anisotropy correction has a special influence on cross-borehole measurements results.

The program restores the average velocity value and Vxy/Vz ratio in a constant cell velocity mode. To calculate anisotropy, choose the *Invert Anisotropy* function in the Options/Inversion tab in the main program menu.

You should be careful when correct the average velocity and anisotropy parameters simultaneously because it gives an additional degree of freedom (gain equivalence) to the solution. So it is better to use this mode in case if you are sure that there are anisotropic rocks in the section.

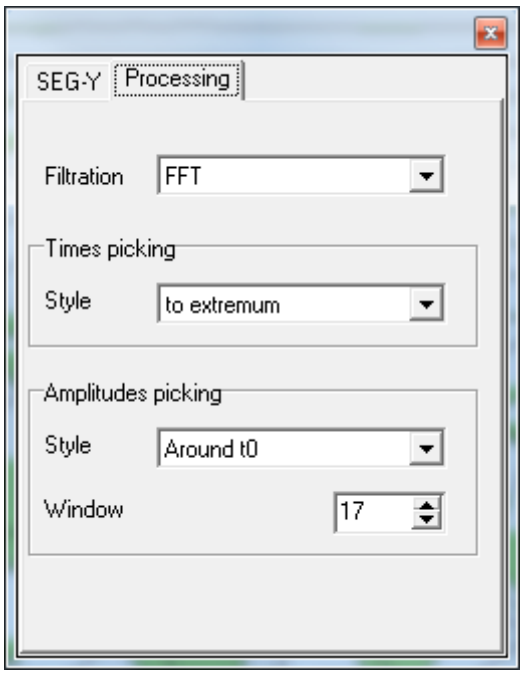
Attenuation tomography mode

Attenuation tomography module is designed to restore the Q medium attenuation parameter. To access this module, select the Attenuation option in the Options/Model/Parameter/Attenuation of the program main menu. Then an additional tab Attenuations appears in the Options menu.

The calculation algorithm assumes that the inverse problem was pre-solved for seismic velocities and ray coverage is known.

You input the following parameters - seismic rays paths, velocities section and the first arrivals amplitudes. The amplitudes are automatically determined when the first arrivals hodographs are being picked. It should be noted that for this algorithm, it is more correct to pick for the maximum (modulo) values. If picking is performed on the wave rising edge – the special options for amplitude values adjusting should be used (reduction to the maximum) - collect amplitudes shot/all and settings Amplitude picking.

For proper option working it is important to choose the search window width in which maximum searching will be performed. Search window settings are in the dialog of picking window.



Basic velocity model (ray coverage) is a very important parameter for inversion. Therefore, before starting calculation procedure, make sure that rays cover close section over the area. Amplitude inversion algorithm uses the same basic settings as the tomography on the refracted waves. Additional parameters that must be set before the calculation is the average frequency of the desired signal (Attenuation/Frequency). It can be estimated visually or by average seismogram spectrum. To start the calculation procedure use the button . During the inversion iterative process the Q model parameter will be displayed in the lower window part, observed and calculated amplitudes graphs - at the top.

Model section visualization modes and parameters

Model can be displayed in a form of cells **Options/Model/Block-section** (fig.49A) or smooth interpolated palette **Options/Model/Smooth-section** (fig.49B)

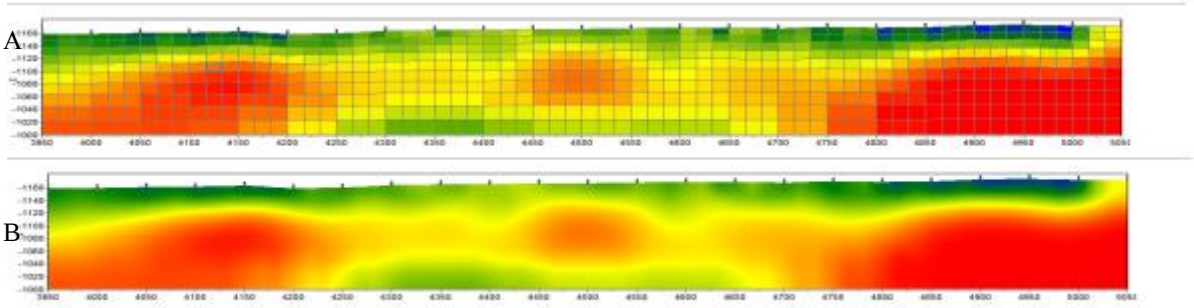


Fig. 49. Model displaying options: ***Block-section*** *(A),* ***Smooth-section*** *(B)*

You can choose the model slice orientation (XZ, YZ, XY) using **Options/Model/Plane** option.

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

Top domain	Display model mesh	Display model mesh
	Display objects border	Display objects border
	Display color bar	Display colour bar
	Setup	Run model parameters setup dialog.
	Zoom&Scroll	Turn on zoom and scroll mode
	Print preview	Print model
	X:Y=1:1	Make scales along the vertical and horizontal axes the same.
Colour bar	Set minimum	Set minimum value of colour bar
	Set maximum	Set maximum value of colour bar
	Automatic	Automatically select minimum and maximum value of colour bar
	Log scale	Set logarithmic scale on colour bar
	Smooth image	Smooth image contours
	Set cursor value	Set current value of the parameter

To view model parameters (red colour) and data (blue color) distribution graph use **Options/Model/Model Histogram** dialog (fig. 50). Minimum and maximum of parameter colour scale can be set by changing vertical lines positions.

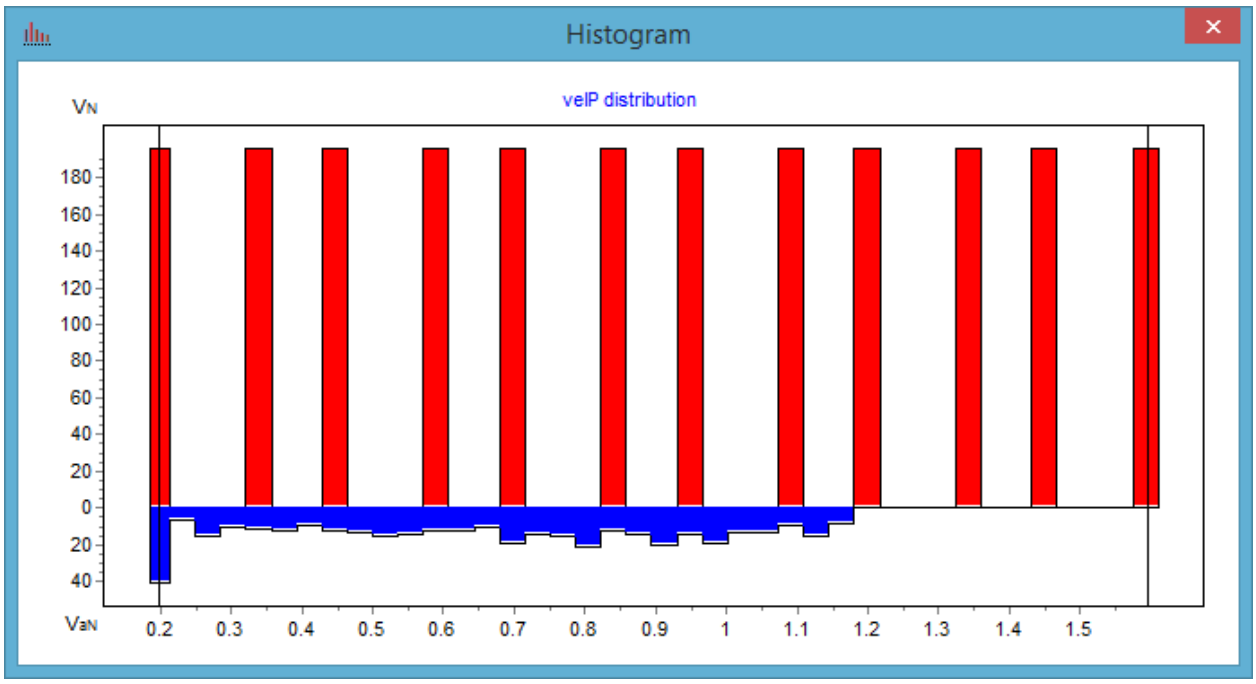


Fig. 50. ***Histogram*** *dialog.*

While moving the cursor in created windows coordinates corresponding to current window axes are displayed in left section of status panel of program main window.

Option «Buffer» in the main program menu

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

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

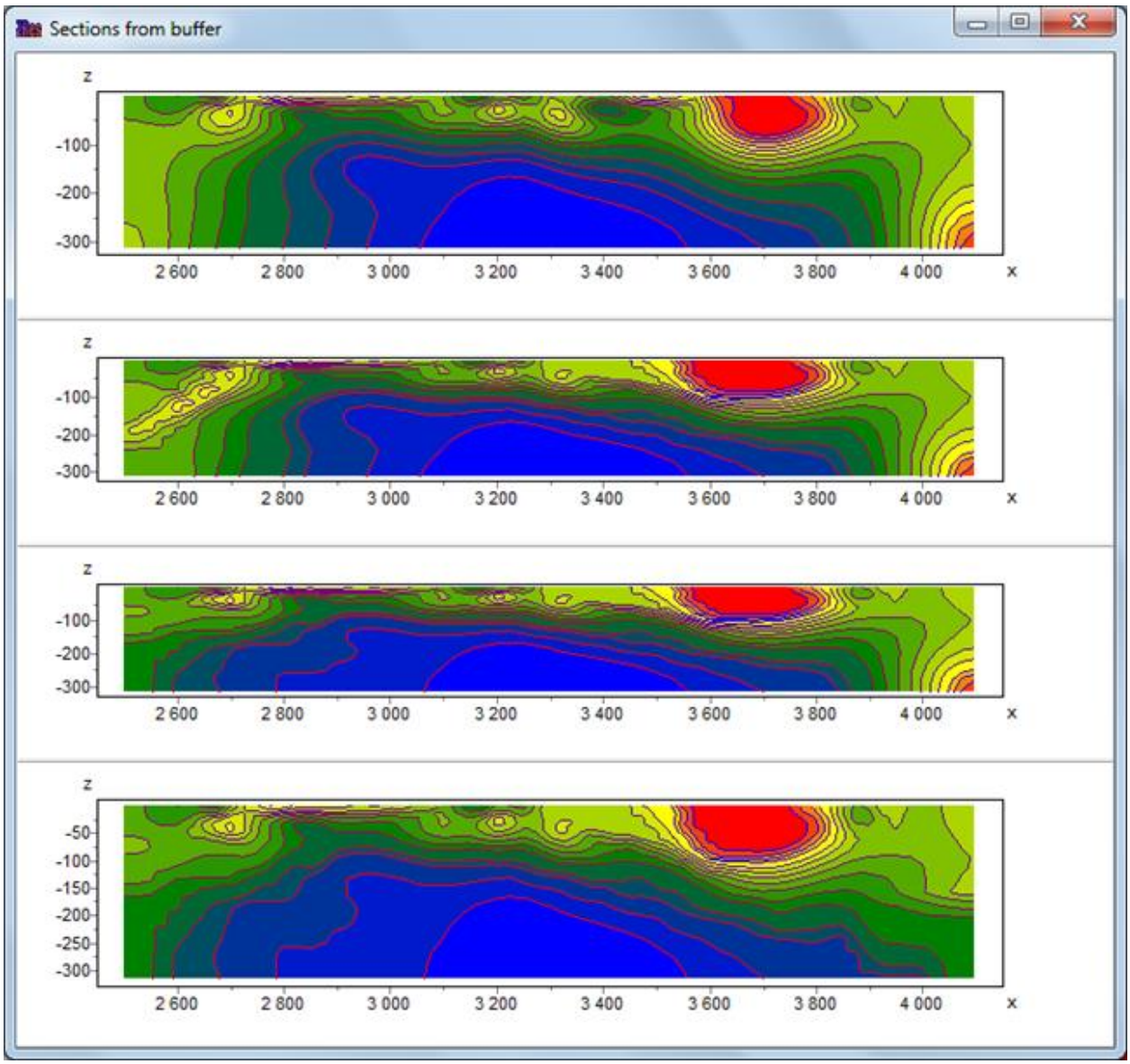


Fig. 51. Sections from Buffer dialog.

3D model parameters setting dialog

3D model serves to display the section in 3D view. The image size is controlled with the mouse wheel. To rotate model, use pressed left mouse button. The button on the toolbar of the program main window serves to set the 3D image parameters, the settings are also available on right mouse button click in the 3D model section, whereupon the **3D Options** appears.

When the button is held down, the following options are available:

3D view settings	Run the 3D model parameters setting dialog
OpenGL settings	3D image editor.
Print preview	3D model print setting dialog.
Plane cut	Choose the model slice orientation.
Model	Set the start model parameters Half-space value – set the value of the parameter for the half-space

	Clear model – clear the current model Clear limits – clear the current parameters limits MinMax auto – set the parameters limits automatically Smooth/raster model – smooth/raster the model
Axises settings	Call the axis parameters setting dialog.
Show	Choose the displayed objects (to choose check the boxes opposite to the needed objects): Cutting plane – show the model borders Survey points – show the measurement points Boreholes – show well log and stratigraphy data Background – show the base layer
Synchronous cut	Change the slice position on the 3D model synchronously with changing the 2D plan using the buttons.
X:Y:Z 1:1:1	Set the same scale for all the axes

The 3D image parameters setting dialog consists of five tabs. The **Opacity** (0-100) option is situated in the bottom part that allows to set the general model transparency. For more delicate model transparency parameter settings use the button (fig. 52).

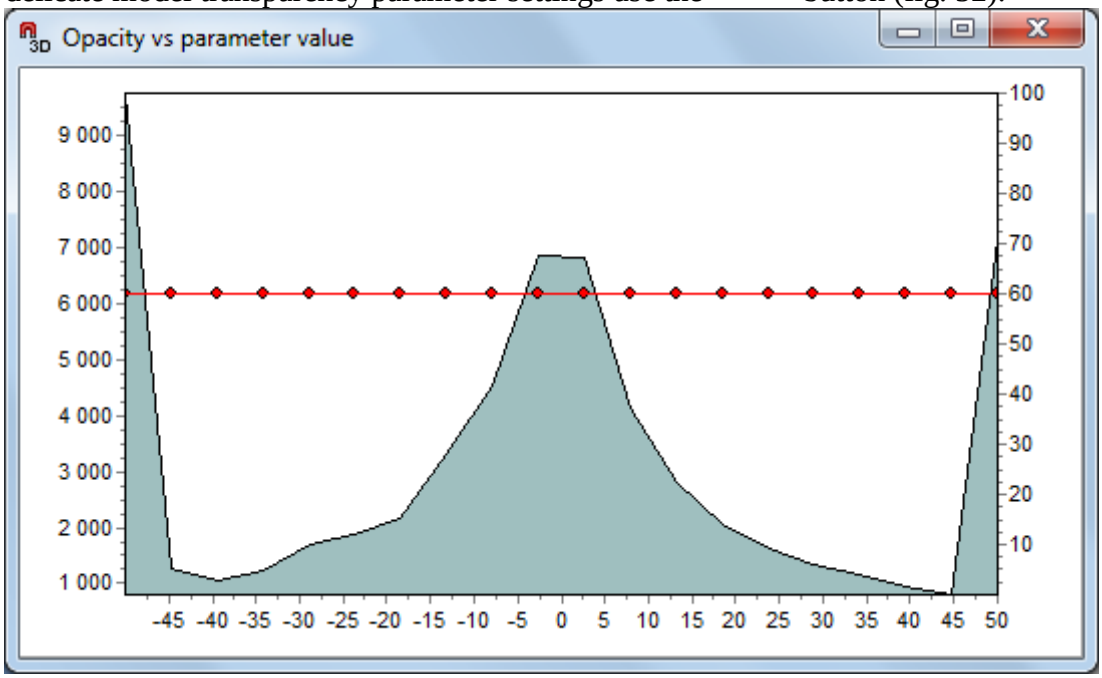


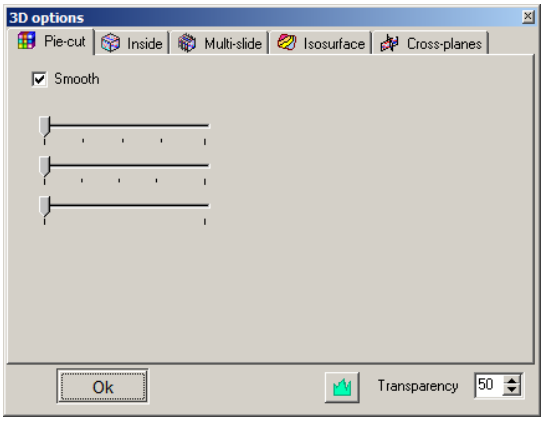
Fig.52. Opacity vs parameter value dialog window.

There is a filled graph in the **Opacity vs parameter value** window which shows the current parameter distribution. Red graph shows the transparency value depending on the parameter. The background medium parameter value is usually a maximum on the distribution graph. If it must be shown only anomalous objects, the transparency of those which are close to background medium is being decreased. Use mouse to change the transparency profile.

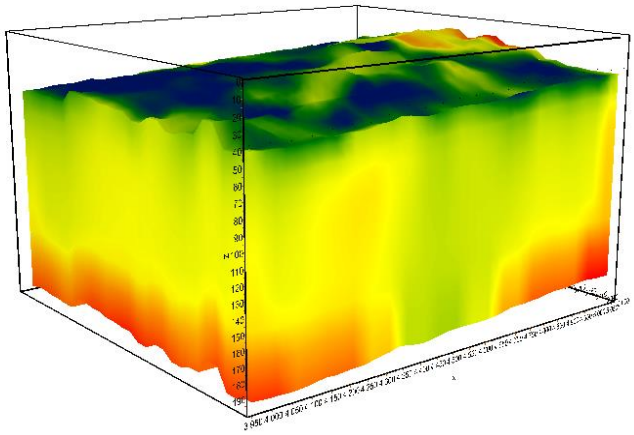
The **Pie-cut** tab – construct a model in the form of opaque parallelogram limited by user-selected cross-sections (fig.53).

The **Smooth** option activates the model constructing regime with continuous interpolative filling. Three sliders set the positions of cutting-off borders for the model.

A



B



C

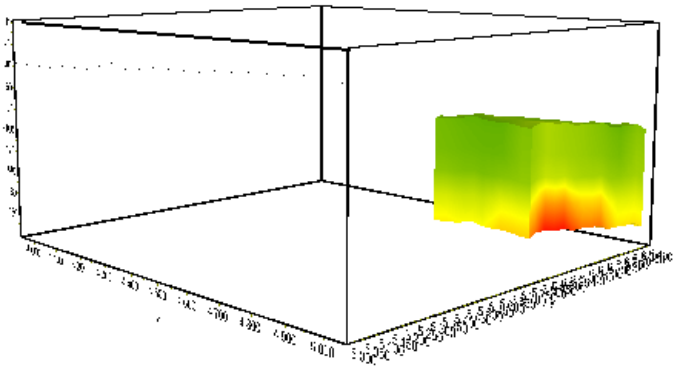


Fig. 53. Dialog window 3D options, Pie-cut tab (A) and examples of the corresponding model visualization (B, C).

Inside tab – fully built the model of semi-transparent blocks (fig. 54). The slider in the top part sets relative cell size from 0 to 100 percent. The unnecessary blocks are cut off in the **Filters** section. **Hide half-space** option forbids drawing of the blocks which's values are equal to those of the half-space (convenient to use in the modelling regime). In case of inversion results displaying it is more suitable to use **Opacity** dialog settings.

A

B

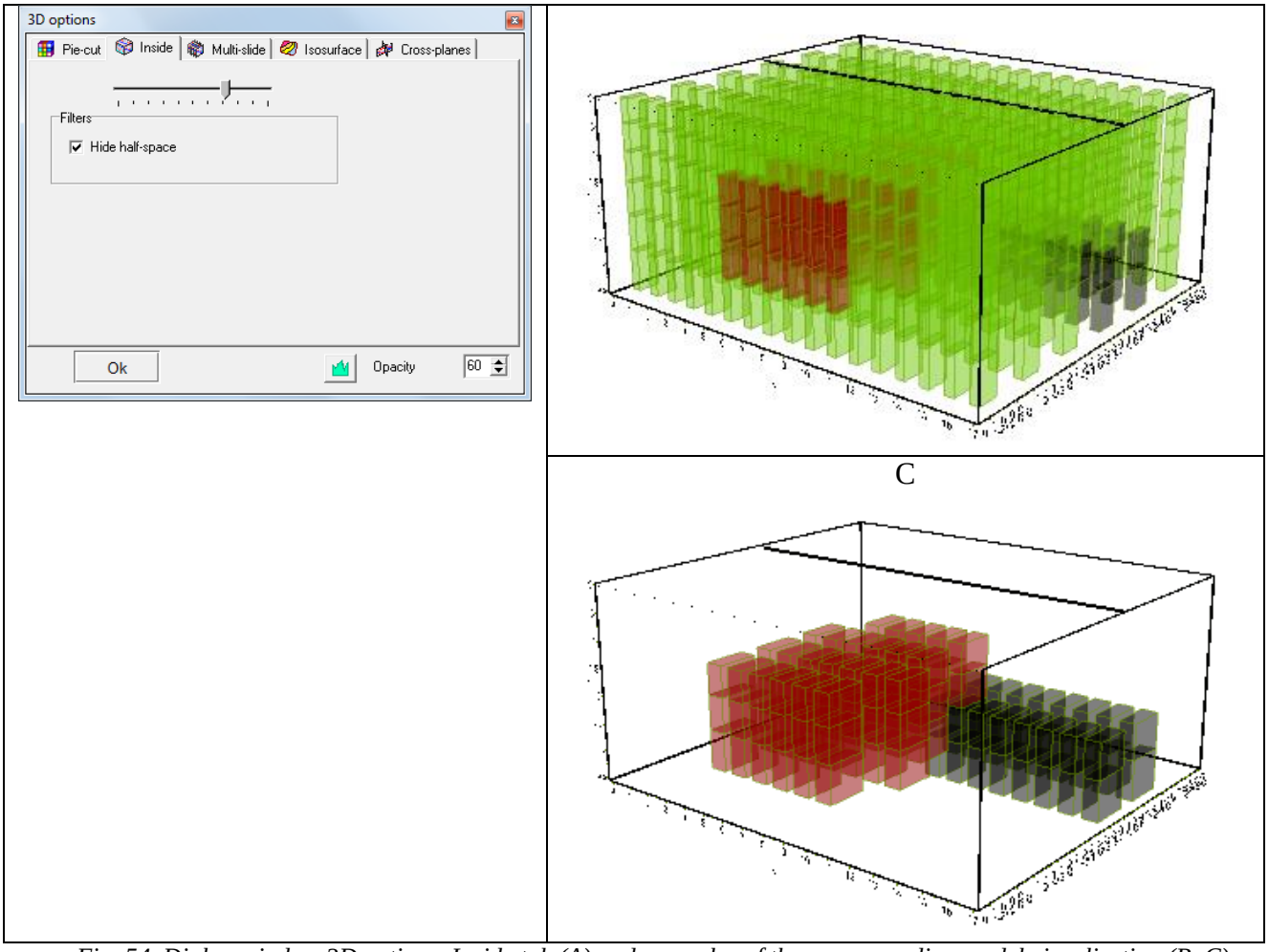


Fig. 54 Dialog window 3D options, Inside tab (A) and examples of the corresponding model visualization (B, C).

Multi-slide tab – shows a stand of semi-transparent model slices along one of the directions (fig. 55). **Smooth** option includes regime of the model graphical construction with a continuous interpolative filling. **Automatic** option includes an automatic selection of the stand direction depending on the angle of vision.

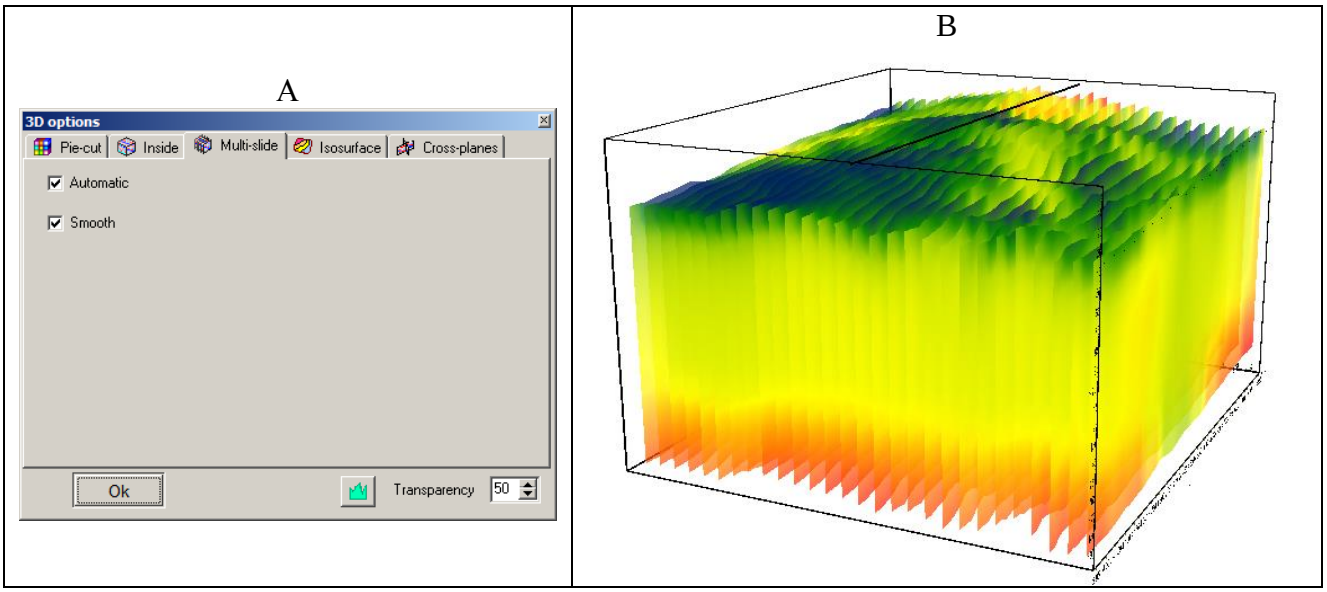


Fig. 55. Dialog window 3D options, Multi-slide tab (A) and an example of the corresponding model visualization (B).

Isosurface tab – displays three-dimensional isosurfaces of the section parameters (fig. 56). Three sliders determine the values upon which the isosurfaces will be built. The **Smoothness** slider determines a smoothness degree of the isosurface. **VolRender** option activates the displaying regime for a semi-transparent fog which's colour is determined by the model parameters.

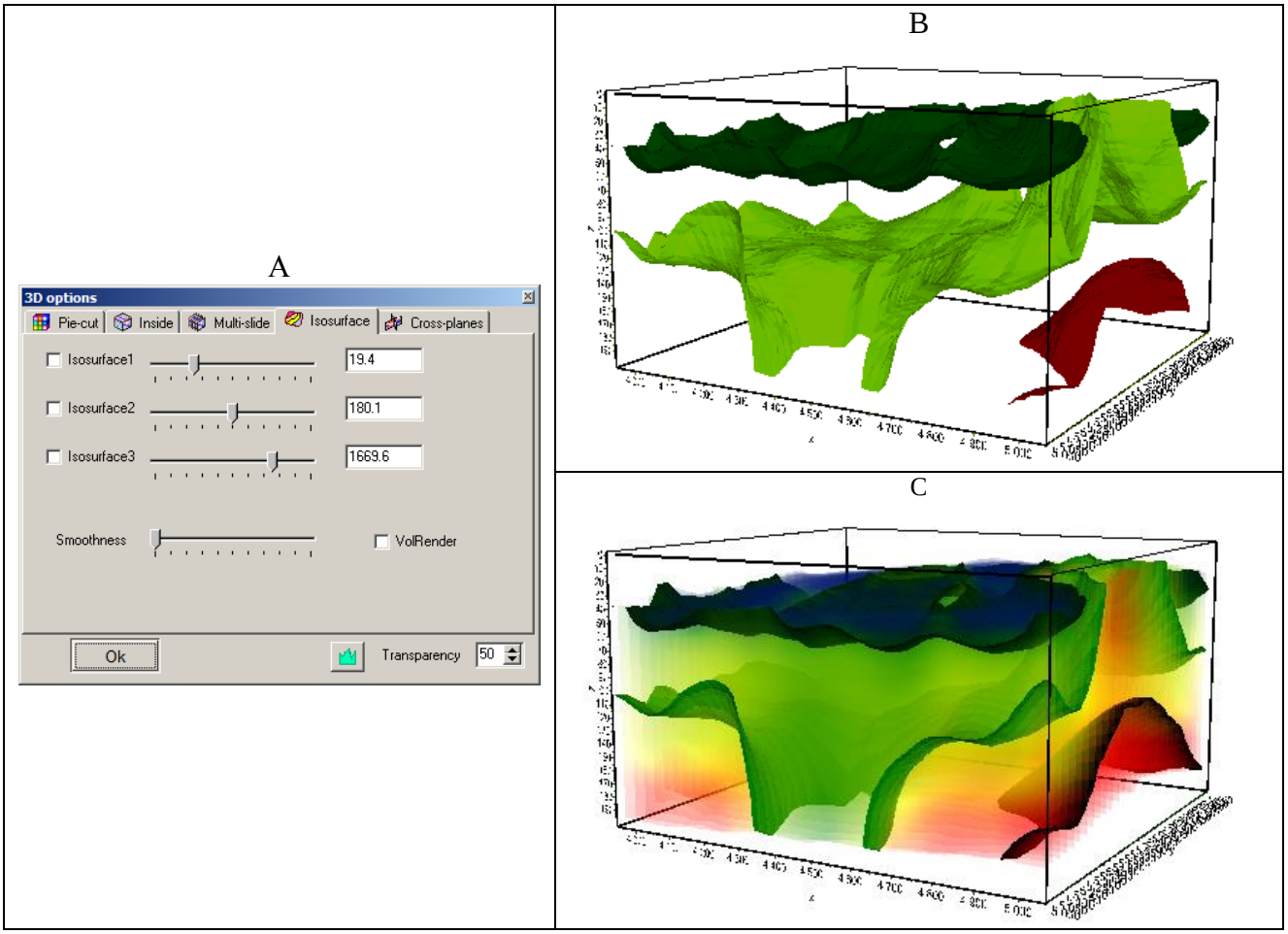


Fig. 56. Dialog window 3D options, Isosurface tab (A) and examples of the corresponding model visualization (B, C).

Multi-planes tab – displays three semi-transparent mutually intersecting planes (fig. 57). **Smooth** option activates the model graphical construction regime with a continuous interpolative filling. Three sliders determine the position of the model intersection planes.

A	B

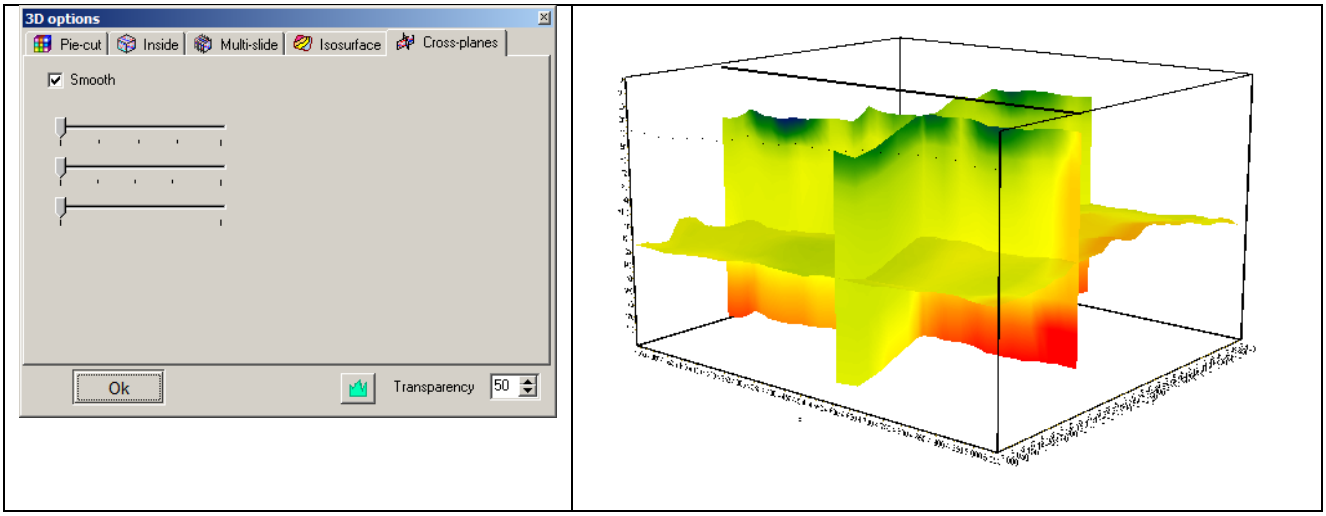


Fig. 57. Dialog window 3D options, Multi-planes tab (A) and an example of the corresponding model visualization (B).

Model editor

Geoelectric medium creation is performed in model editor (bottom graphic section of program window in **block-section** mode).

Model editor serves for changing certain cell parameters using mouse. To the right of model edit region there is a colour bar that connects colours and resistivity values. Right click on the scale to select current value; this value will be displayed below colour bar.

Work with model cells is similar to raster image editing in graphics editor. When you move cursor in model domain coordinates and parameters of active cell are displayed in the bottom status panel of program main window. Current active cell is highlighted by rectangular – cursor. Selected or fixed cell are marked by white or black dots pattern.

Right click in model editor domain to run context menu that contains the following options:

Display cell setup	Run cell parameters setup dialog
Cell to cursor value	Use active cell parameter as current value
Edit mode	Run Edit mode
Selection\Free form selection	Highlight set of cells within edit region using mouse. Field is limited by user.
Selection\Rectangular selection	Highlight set of cells within edit region using mouse. Field has rectangular form.
Selection\Elleptical selection	Highlight set of cells within edit region using mouse. Field has elliptic form.
Selection\Magic wand	Highlight set of cells within edit region using mouse. Active cell and adjacent cells whose parameters are close to its parameter are highlighted. Proximity is specified in model parameters setup dialog.
Selection\Remove selection	Delete selected
Clear model	Clear current model

Operating with model

Use mouse to work with model.

Left click on cell to set its parameter current value.

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

In order to efficiently create a model there are a few cell selection modes in the program: rectangular, elliptical and free form selections or by certain parameter value. Right click in model edit region to run appropriate options.

Click on the cell with CTRL button pressed to drag selected set of cells within edit range using mouse. While dragging the selection with left button pressed content of selected cells copies to new domain. While dragging the selection with right button pressed content of selected cells is cut and pasted to new domain.

Parameter value can be also assigned to selected cells using cell parameter setup dialog **Cell setup** (fig. 58).

Cell parameters setting dialog

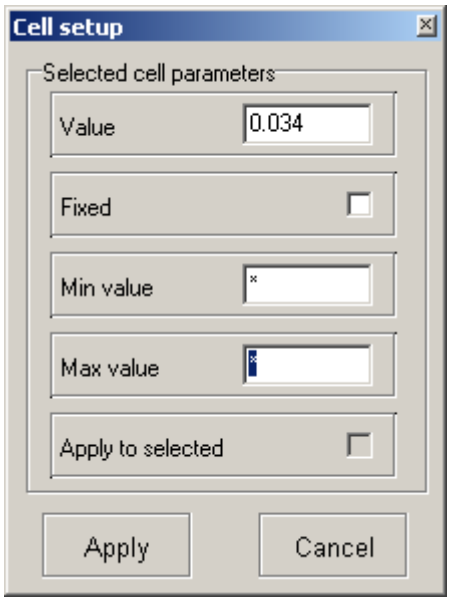


Fig. 58. ***Cell setup*** *dialog.*

This dialog serves for selecting cell parameters or highlights it.

Value – sets cell parameter value.

Fixed – fixes or frees cell parameter.

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

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

It should be noted that changing modes (resistivity/polarizability) does not change selected domain.

The main purpose of mathematical modeling is signal level estimation and acquisition geometry resolution evaluation. Press button in the tool bar to calculate apparent parameters response in specified geoelectric model (forward solution).

It is possible to regenerate starting model, that is to solve inverse problem using data calculated from current model, if theoretical signals are saved with filter - **Zond calculated data**, and then open as observed data.

Saving interpretation results

Profile interpretation result is hold in «ZONDST3D» file format (extension *.st3). Field data, relative measurement weights, and current subsurface model are saved in this file. Data from the file is used for further load and subsurface model creation.

The interpretation result can be saved by pressing the button on the toolbar or the corresponding menu item **File/Save file**.

Project file	Save observed data and current subsurface model
Project with calculated data	Save calculated data and current subsurface model
Observed	Save observed data
Calculated data	Save calculated data
XYZV	Save table file *.dat with observation points coordinates and calculated values of the parameter
XYZ	Save table file *.dat with observation points coordinates

Data import and export

Usage of graphical base layers for 2D and 3D models

Integration of geophysical methods and accounting a priori information are the most powerful methods to enhance interpretation data quality. There are a few ways of a priori information visualization in the program. **Options/Import/Export** option allows loading different geological and geophysical information including profile measurements such as graphs, models built in other Zond programs, graphic image as a base for section (for example, geological or seismic section) (fig. 59).

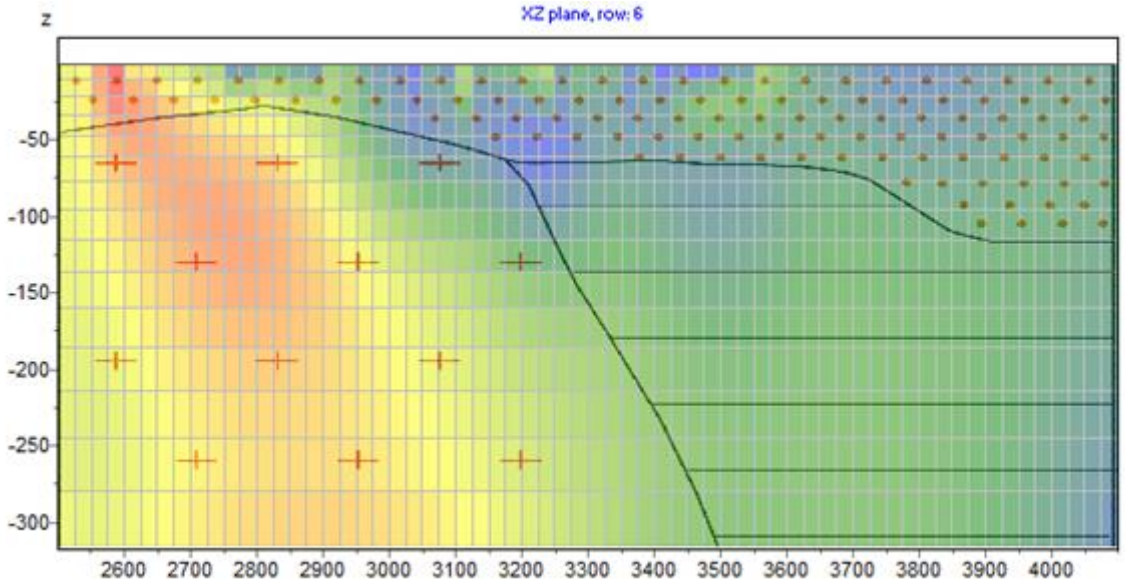


*Fig. 59 An example of usage **Background 2D** function*

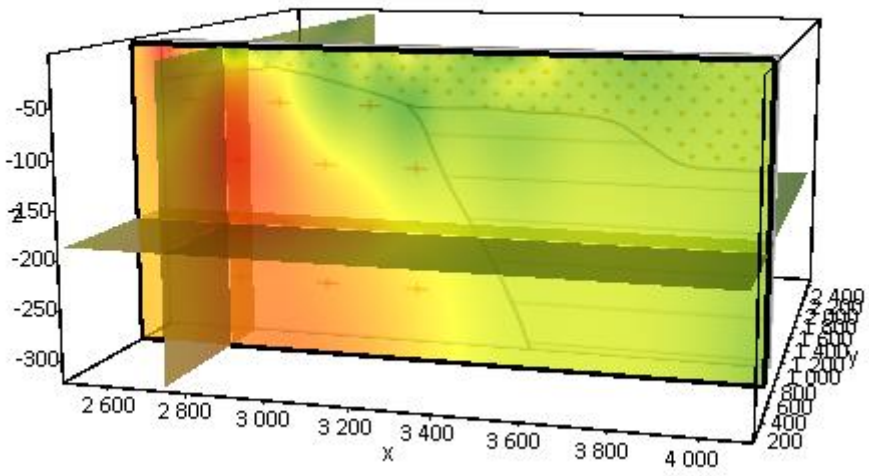


*Fig. 60 An example of usage **Background 23** function*

Using **Save rotations** option you can save pictures of the model rotation about Y axis in the chosen direction of the «camera».

The pictures of model sections in the XZ plane in the chosen direction of the «camera» can be saved using **Save slides** in the model displaying regime **Multi-planes.**

To compare the results of different methods, it is convenient to use **Import Mod2D's** and **Import mod3d** functions. These options allow uploading models saved in other Zond programs as MOD3D and MOD2D files into the current model as a current model.

Different options for data and resultant model export are described in *«Saving interpretation results».* For the further geological interpretation and preparation of report graphics there is an opportunity to save the model into three-dimensional dat-file or the model slices in the XZ plane into 2D dat-files. The model can be saved as a bitmap (**Save slides)** of definite resolution and size using the **Output settings** dialog.

Export image setting dialog

Output settings dialog allows adjusting vertical **Vertical scale** (in meters per sm), horizontal scale **Horizontal scale** (in meters per sm), image resolution Print resolution (in DPI) and font size **Font size**.

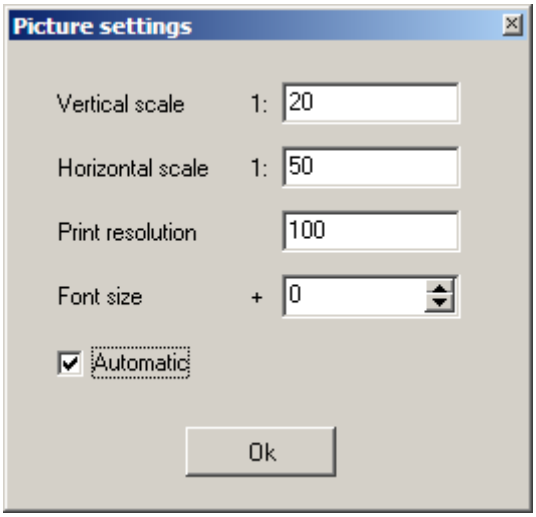


Fig. 61. ***Picture settings*** *dialog.*

These settings are applied to model saved in BMP format , only if **Automatic** option is activated. Otherwise the same image that is displayed on the screen will be saved.

Borehole data creation

If you have borehole data and want to create a stratigraphy file select **Options/Boreholes/Create/Edit borehole data** in the main program menu. The dialog box **Add borehole data** appears (fig. 62).

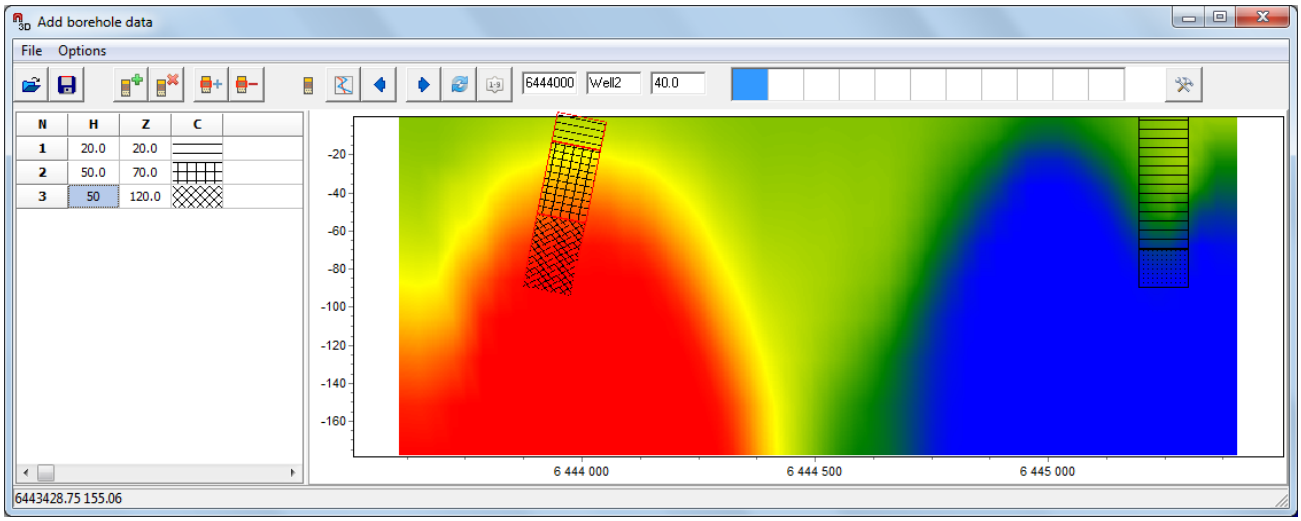


Fig. 62 Lithology and logging file creation dialog ***Add borehole data****.*

Dialog box toolbar contains the following buttons:

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

There are also windows of setting the horizontal well coordinate (the line start offset) – **horizontal position**, name of the well – **title**, and the inclination angle – **angle** on the main panel. The horizontal coordinates of the wells are set in meters. Small empty boxes are intended

to create the filling set. To change the horizontal coordinates, use the program main menu function **Options/Boreholes/Edit position**, and then the **Locations** dialog window will appear (fig. 63) where you can set or upload coordinates.

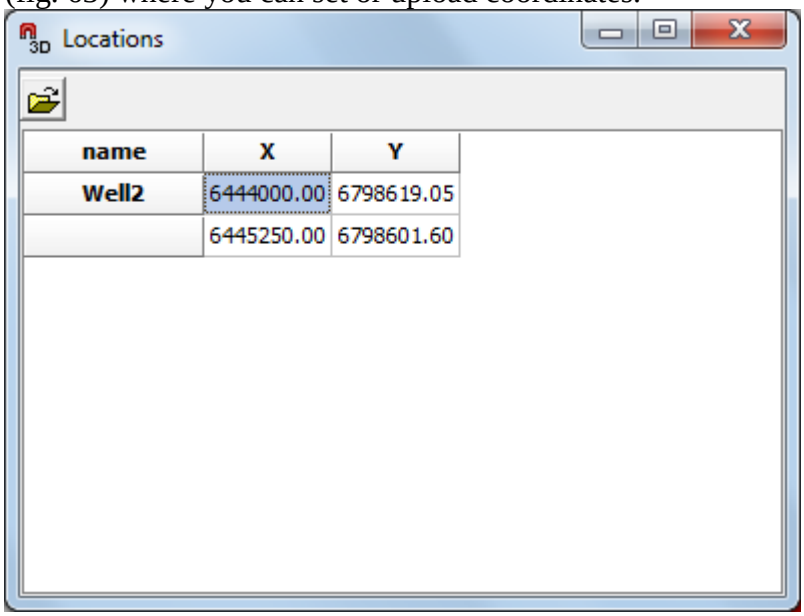


*Fig. 63 Well creating dialog window **Locations**.*

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

To begin creating a lithology file click the button on the toolbar. Then a new table appears in the **Data Window**. To set number of layers, click the button . Then it is necessary to edit the table: set thickness or bottom depth for every layer, and also choose the filling type according to the lithology. To call filling dialog **Pattern Color Editor**, double click the left mouse button in column **C** of the data window (fig. 64). The program offers a wide choice of lithology fills. The option **Color** allows choosing the fill color.

Fig. 64 Fill editor window

After completing entering borehole data, press the button and the borehole appears in the graphics window. After that it is necessary to set the horizontal and vertical borehole

coordinates on the toolbar in kilometers, after that the borehole will be displayed in accordance with its coordinates. In the graphs window active borehole is displayed in red.

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

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

After saving the data file, several data files will be created: ***.crt** - module project that can be loaded into the program and ***.txt** - file for each borehole, the names correspond to the horizontal and vertical coordinate. More about the format of lithology file.

To add borehole data, use **Options/ Borehole/Load borehole data.** Borehole data will be displayed on both model section and in the 3D model area (fig. 65).

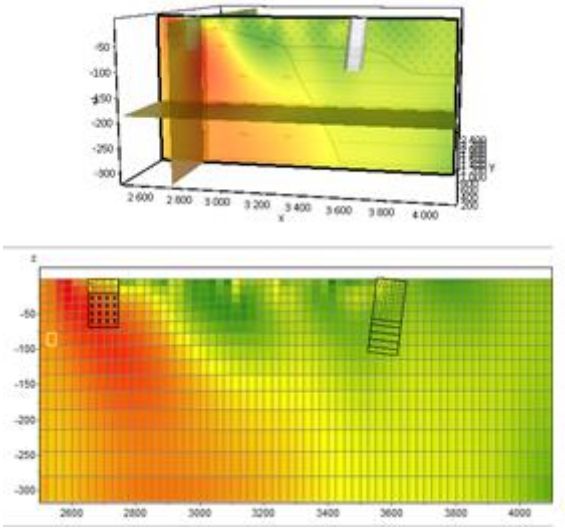


Fig. 65 Lithology data display in the section editor.

To load the available logging data of boreholes, prepare a file format described in the section «Logging and lithology data file format», and load it using the button .

Lithology and logging data file format

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

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

Logging data sample-file is given below:

0.5	118.3035394	0	0
1	126.9002384	0	0
1.5	123.4170888	0	0
2	116.1519574	0	0
2.5	117.240884	0	0

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

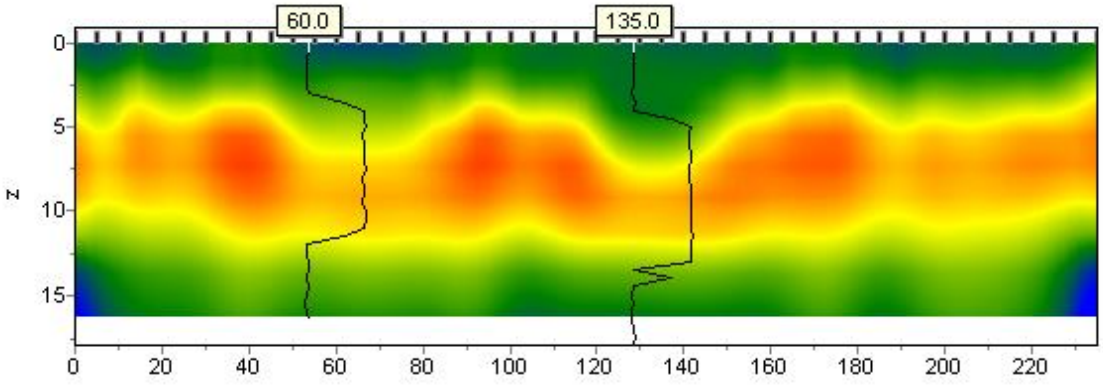


Fig. 66. Model with plotted well logs.

The following structure is used to create lithology data file:

First column contains lithological layer depth (from surface). Second column is filled with zeroes. Third column defines layer colour for visualization, forth – type of pattern.

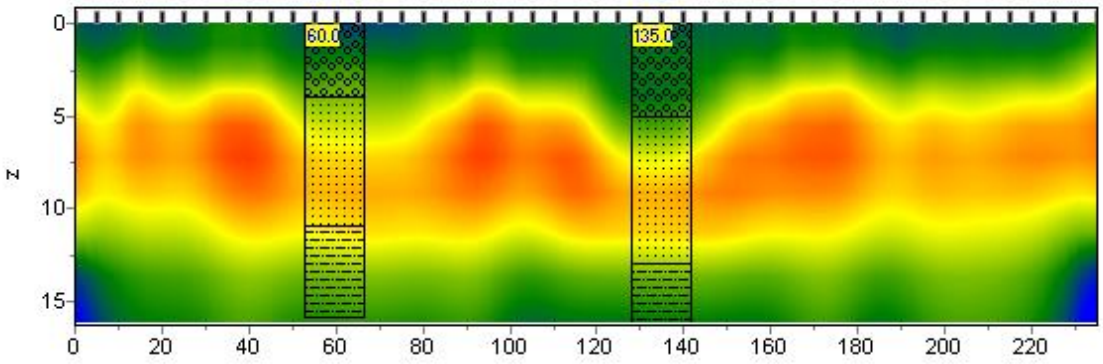


Fig. 67. Model with plotted lithologic columns

First 23 patterns for lithological column creation are given below (fig.68).

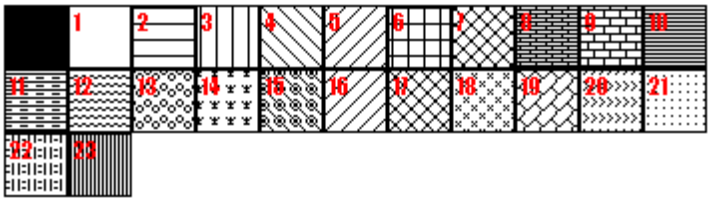


Fig. 68. Types of lithologic pattern.

Lithological data sample-file is given below:

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

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

2280.txt First line – logging or lithology data file
скв2280 Second line – Well name (is displayed on well)
18 2 2 1 0 1 0 0 Third line contains control parameters -
Data record 18 – well coordinate on profile.
2 – image width (in percents to profile length, usually 1 - 20).
2 – type of data visualization 0 - 3.
0 - logging data (as graph);
1 - logging data (interpolated colour column), section colour scale is used for visualization;
2 - lithologic column;
3 - logging data (colour column), colours for data visualization correspond to model colour scale, column colours are selected in compliance with model colour scale;
1 – Logging data normalization parameter 0 - 2.
0,1 – the same minimum and maximum is used for all data;
1,2 - subtract average value from every well log;
0 - Logging method index (if different logging methods are displayed indices of all methods should be specified) 0 – n-1, where n – number of methods.
1 – Plot colour.
0 – Data scale is logarithmic 0 or linear 1.
0 – Vertical well shift relative to the earth's surface.

3246.txt Description of the following well on profile
скв3246
102 2 2 1 0 1 0 0

Appendix 1. Palette settings

Dialog can be used for setting object palette of the program and is run by clicking the button **Palette** (fig. 69). Dialog allows selecting one of the default palettes (forward and inverse rainbow, grayscale, etc.) or create a custom scale. To add a slider on the scale, click the right mouse button with pressed Ctrl. To remove a slider, press the key Delete. It is also possible to save a custom palette. To do this, click the button . To load existing one, click the button .

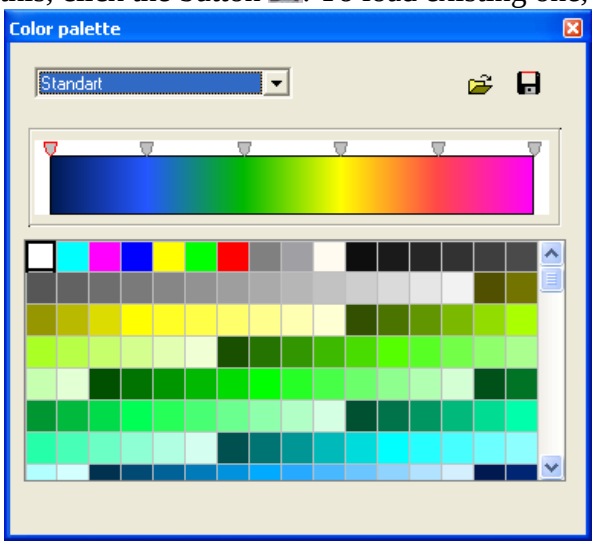


Fig. 59 Palette settings dialog.

Appendix 2: Graphs set editor

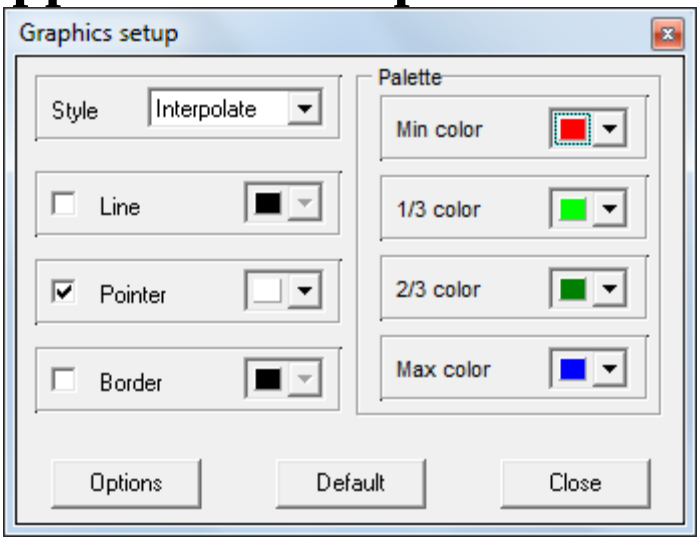


Fig. 70 Graph set customization dialog.

Graphs set editor serves for colour adjustment of graphs set.

Option **Style** defines algorithm of graphs colour palette specification.

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

Option **Line** sets colour for graph connecting lines. If this function is deactivated, palette colour is used, otherwise colour specified in **Line** field is used.

Option **Pointer** sets colour for graph point's colour fill. If this function is deactivated, palette colour is used, otherwise specified in **Pointer** field colour is used.

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

Button **Options** runs graphs setup dialog.

Button **Default** returns graphs default settings.

Appendix 3: Graphs editor

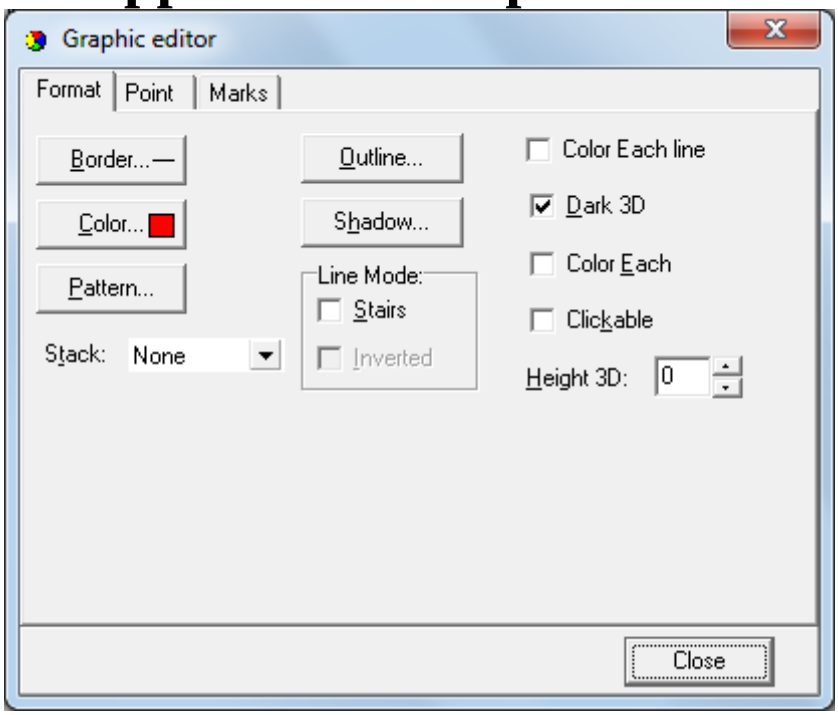


Fig. 71 Graph setting dialog.

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

Tab **Format** contains connecting line settings.
Button **Border** runs connecting line parameters setup dialog.
Button **Colour** runs colour setup dialog.
Button **Pattern** runs filling parameters setup dialog.
Button **Outline** runs graph connecting line setup dialog.
Button **Shadow** runs shadows setup dialog.

Tab **Point** contains plot point settings.
Option **Visible** is used to show/hide plot points.
Option **Style** sets point shape.
Option **Width** sets point width in display units.
Option **Height** sets point height in display units.
Option **Inflate margins** defines if image size is zoomed in according to point size or not.
Button **Pattern** runs point's colour fill setup dialog.
Button **Border** runs point's outline parameters setup dialog.
Button **Gradient** runs point's gradient colour fill setup dialog.

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

Tab **Style**.

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

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

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

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

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

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

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

Option **Length** sets arrow length.

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

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

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

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

Button **Colour** runs frame background colour selection dialog.

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

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

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

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

Option **Size** rounds frame corners.

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

Tab **Text**:

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

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

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

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

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

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

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

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

Appendix 4: Legend editor

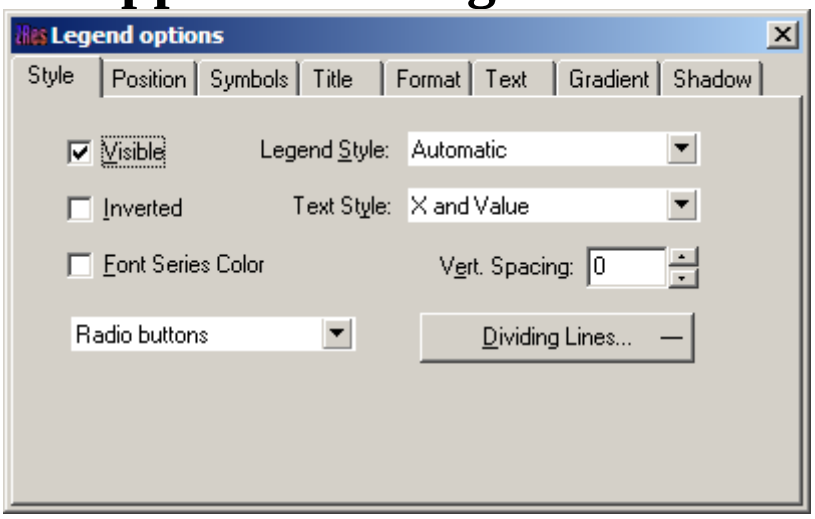


Fig. 72. Graph legend setting dialog.

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

Tab **Style** contains settings of legend display, allows choosing data label format and showing boundaries between legend labels and so on.
Tab **Position** serves for choosing legend position relative to graphs plan.
Tab **Symbols** sets legend symbols display parameters.
Tab **Title** specifies legend name and allows adjusting its format.
Tab **Text** serves for adjusting legend label format.
Tabs **Format**, **Gradient** and **Shadow** contain settings of legend window, its gradient fill, and shadow.

Appendix 5: Pseudosection parameters setup dialog

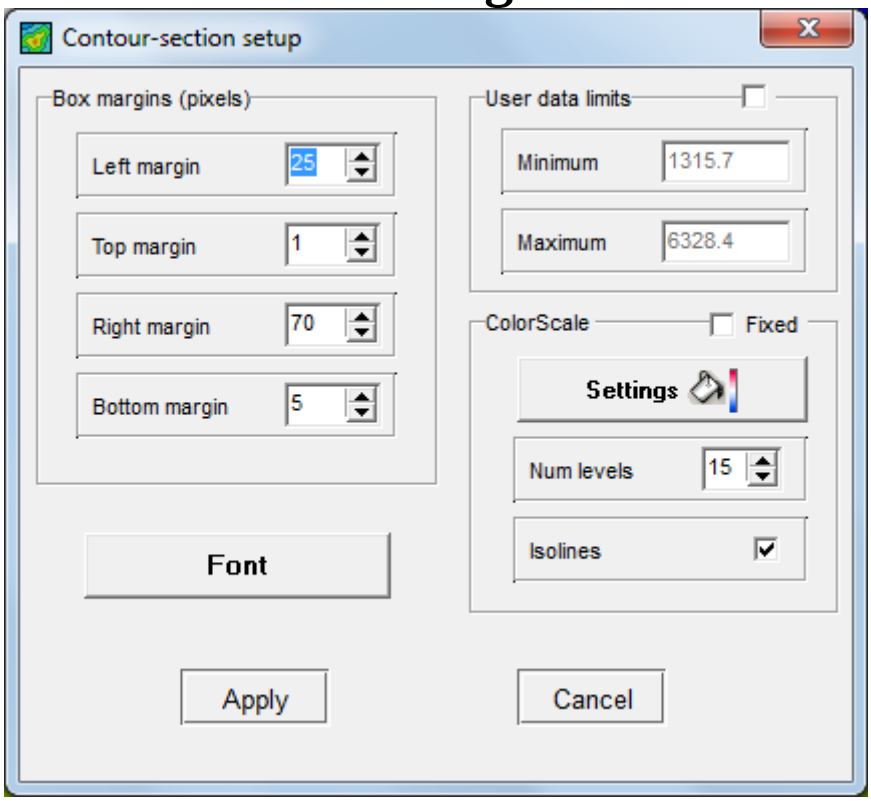


Fig. 73 Contour-section parameters setting dialog.

This dialog serves to set the contour-section parameters.
Field **Box margins**
Left margin – sets image indent (in pixels) from window left edge.
Right margin – sets image indent (in pixels) from window right edge.
Top margin – sets image indent (in pixels) from window top edge.
Bottom margin – sets image indent (in pixels) from window bottom edge.

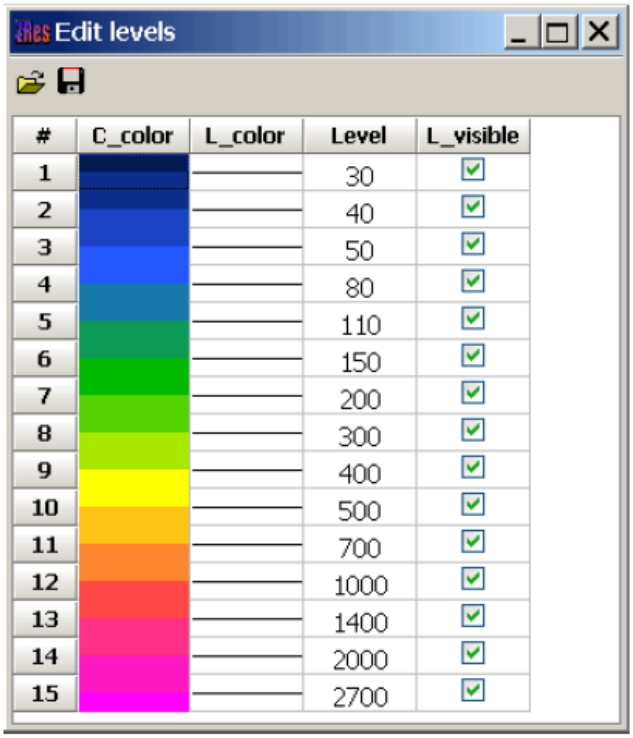
Field **User data limits**
Option **User limits** specifies whether minimum or maximum data values or values from **Minimum** and **Maximum** filed are used for setting contour intervals.
Field **Minimum** sets minimum value to specify contour intervals.
Field **Maximum** sets maximum value to specify contour intervals.

ColorScale section allows operator of creating palette (check the box to use these settings).

Num levels section – sets a number of contour intervals. Contour intervals are set with uniform linear or logarithmic spacing depending on the data type.
Isolines option – tell the program whether to draw contours.

To configure filling parameters, use the **Settings** button. After clicking this button there will be the **Edit Levels** dialog (fig. 74). The dialog is presented as a table with the following columns. *C_color* – colour scale for filling. To edit the colour scale, right-click on the name of the column and you will see the Color palette dialog. To edit a certain colour, right-click on this colour. *L_color* – colour scale for contours. Editing is conducted similarly to the colour scale for filling. When right-clicking on the column name, the Automatic dialog appears, there you can set

maximum and minimum value for the contours and also specify logarithmic contour spacing.
Column *L_visible* – allows to deactivate one or more contours, deselect the check box of the corresponding value.



Pic. 74 Dialog window Edit levels

The **Font** button is intended for the setting of the filling legend font parameters. After clicking the **TeeFont Editor** dialog will appear.
The **Font** button calls the font setting dialog for the filling legend.
The **Outline** button runs the setting dialog for the letters outline.
The **Inter-char** spacing option sets the letter spacing for text.
The **Gradient** button runs the setting dialog for gradient text filling.
The **Outline gradient** option specifies where to apply gradient text filling, on the outlines or the letters interior.
The **Shadow** button runs the setting dialog for the look of the text dropping shadow.

Appendix 6: Axes editor

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

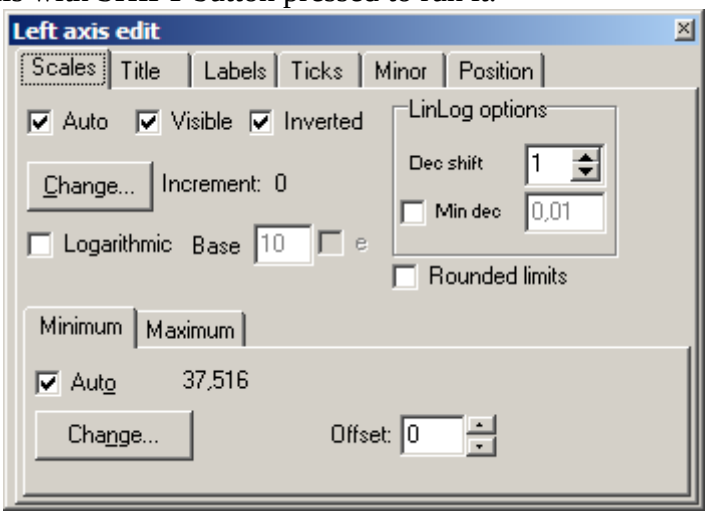


Fig. 75 Axes editor window

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

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

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

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

Option **Inverted** defines axis orientation.

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

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

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

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

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

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

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

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

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

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

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

Tab **Style**:

Option **Title** sets axis header text.

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

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

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

Tab **Text**:

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

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

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

Button **Gradient** runs gradient fill setup dialog for header text.
Option **Outline gradient** specifies if gradient fill is used in outline or interior of letters.
Button **Shadow** runs axis header shadow setup dialog.

Tab **Labels** contains options for axis label adjustment.
Tab **Style**:
Option **Visible** shows/hides axes labels.
Option **Multiline** is used for setting multiline axes labels.
Option **Round first** rounds first axis label.
Option **Label on axis** hides labels that go beyond axis.
Option **Alternate** arranges labels in two lines.
Option **Size** sets axis label indent. If 0 value is specified it is selected automatically.
Option **Angle** sets label rotation angle.
Option **Min separation %** sets minimum percentage label spacing.
Tab **Text**:
Button **Font** runs label font setup dialog.
Button **Outline** runs dialog for label letters' outline adjustment.
Option **Inter-char spacing** sets letter spacing in label text.
Button **Gradient** runs label gradient fill setup dialog.
Option **Outline gradient** specifies whether gradient fill is used in outline or interior of letters.
Button **Shadow** runs label shadow setup dialog.

Tab **Ticks** contains options for axis main ticks adjustment.
Button **Axis** runs axis line setup dialog.
Button **Grid** runs line setup dialog for main ticks' grid.
Button **Ticks** runs external main axis tick setup dialog. Option **Len** sets its length.
Button **Inner** runs internal main axis tick setup dialog. Option **Len** sets its length.
Option **Centered** centers grid of axis ticks.
Option **At labels only** displays main axis ticks only if axis labels are present.
Tab **Minor** contains options for axis intermediate ticks adjustment.
Button **Grid** runs line setup dialog for intermediate ticks grid.
Button **Ticks** runs external intermediate axis tick line setup dialog. Option **Len** sets its length.
Button **Minor** runs internal intermediate axis tick line setup dialog. Option **Len** sets its length
Option **Count** sets number of intermediate ticks between main ones.
Tab **Position** defines axis size and position.
Option **Position %** sets axis indent relative to its standard position on graph (in percent to graph size or in screen units depending on selected option Units).
Option **Start %** sets axis start indent relative to its standard position on graph (in percent to graph size).
Option **End %** sets axis end indent relative to its standard position on graph (in percent to graph size).

Appendix 7: Model parameters setup dialog

Options tab

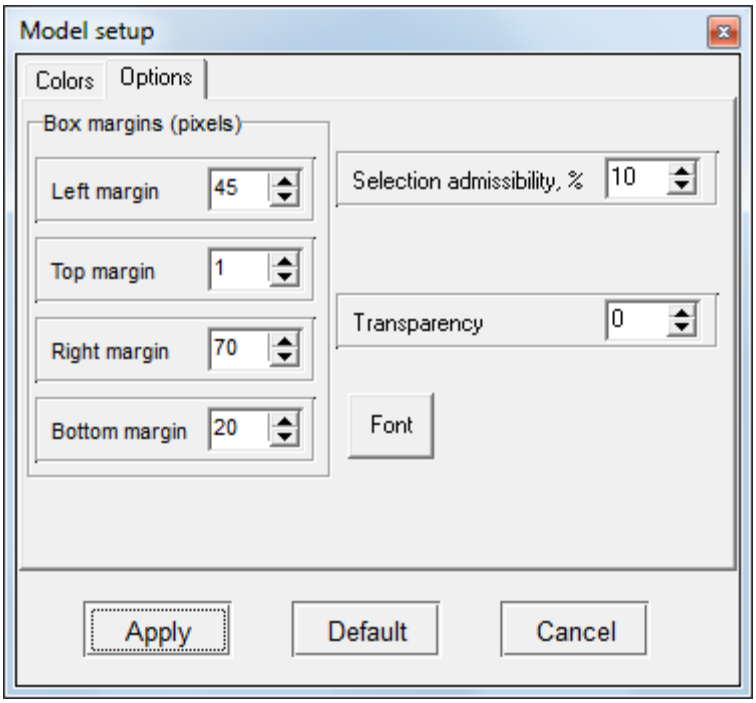


Fig. 76 Setting dialog for the model parameters, **Options** tab.

Field **Box margins**
Left – sets image indent (in pixels) from window left edge.
Right – sets image indent (in pixels) from window right edge.
Top – sets image indent (in pixels) from window top edge.
Bottom – sets image indent (in pixels) from window bottom edge.
Selection admissibility sets threshold value of adjacent cells parameter difference which defines their unification and highlighting together (in Magic Wand mode).
Transparency – sets the transparency values.
The **Font** button runs font setting dialog.
Colors tab.

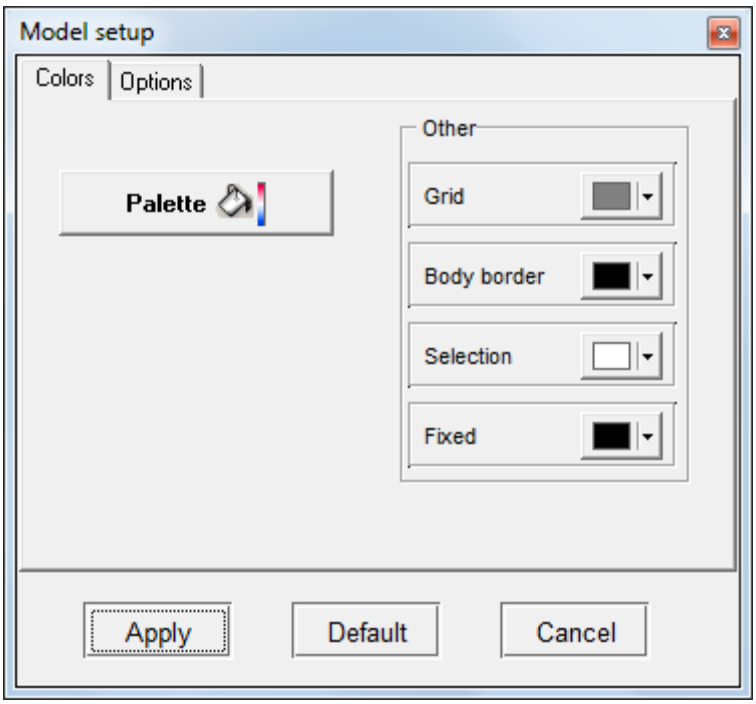


Fig. 67 Setting dialog for the model parameters, **Colors** tab.

The **Palette** button runs the palette setting dialog.
Field **Others**
Body border sets colour of boundary line between adjacent cells if measure of discrepancy between them exceeds specified in **Parameter alteration** option value.
Grid sets mesh colour.
Selection sets mark colour of selected cell.
Fixed sets mark colour of fixed cell.

Appendix 8: 3D model editor

Dialog is available when clicking the button, then you can choose **OpenGL settings** option. The dialog window **TeeOpenGL Editor** will appear (fig. 68).

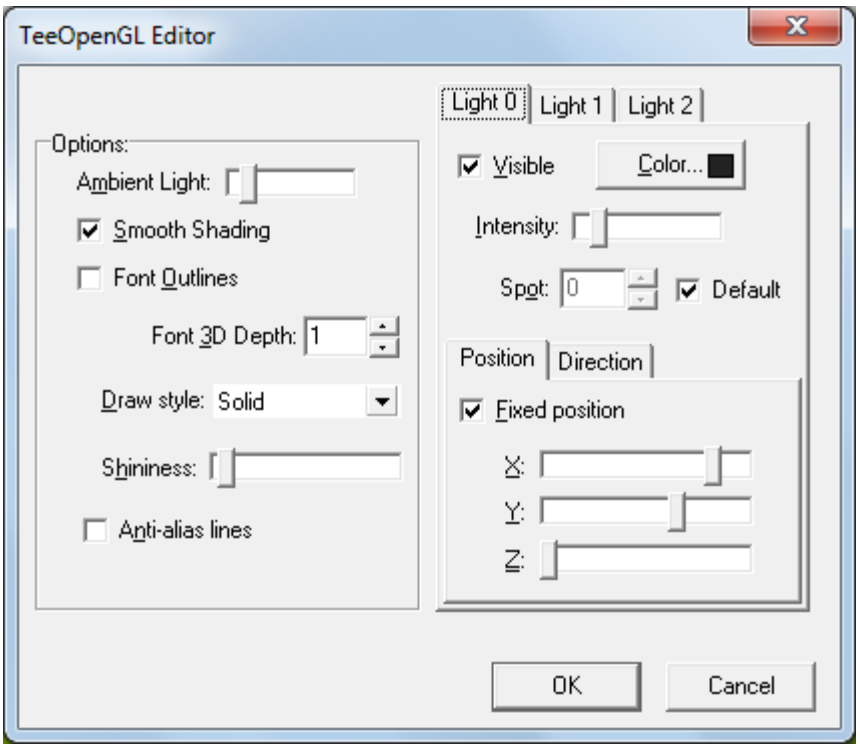


Fig. 78 3D model editor **TeeOpenGL Editor**

The **Options** section is intended to set general image parameters.
Ambient Light – sets brightness of external backlighting.
Smooth Shading – smooths shadows.
Font Outlines – highlights axes titles.
Font 3D Depth – sets font depth.
Draw style – chooses 3D model rendering style (Solid – solid filling, Wireframe – show only frame, Points – show as points).
Shininess – sets brightness.
Anti-alias lines – activates/deactivates smoothing lines.

Light 0, 1 и **2** tabs set the light source parameters.
Visible – activates/deactivates the chosen source.
Color… - sets colour of light source.
Intensity – sets intensity of light source.
Spot – size of source, Default – sets the default value.
Fixed position – sets position of source along axes.
Set direction – sets light direction along axes.

Appendix 9: Print preview dialog

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

When choosing this option by right-clicking on any object of the program, only this object will be printed.

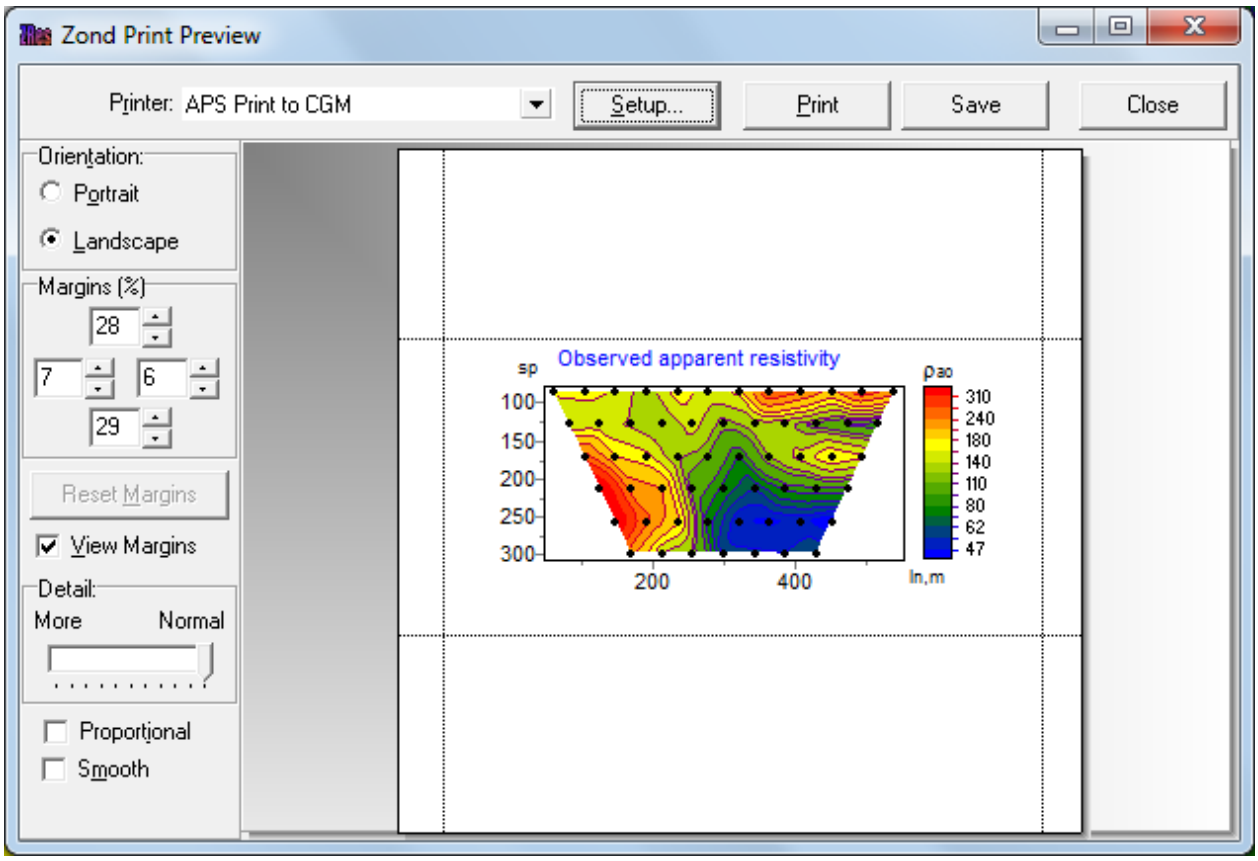
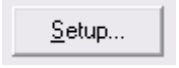


Fig. 79 **Print preview** window

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

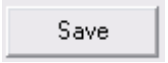
In the main menu of the **Print Preview** window the following buttons are:

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

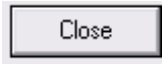


- The button prints settings. In the opening window, it is possible to choose the size and orientation of the sheet, the print properties, the number of pages per sheet and other parameters.

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



- Save in bitmap files.



- Closes the Print preview window.