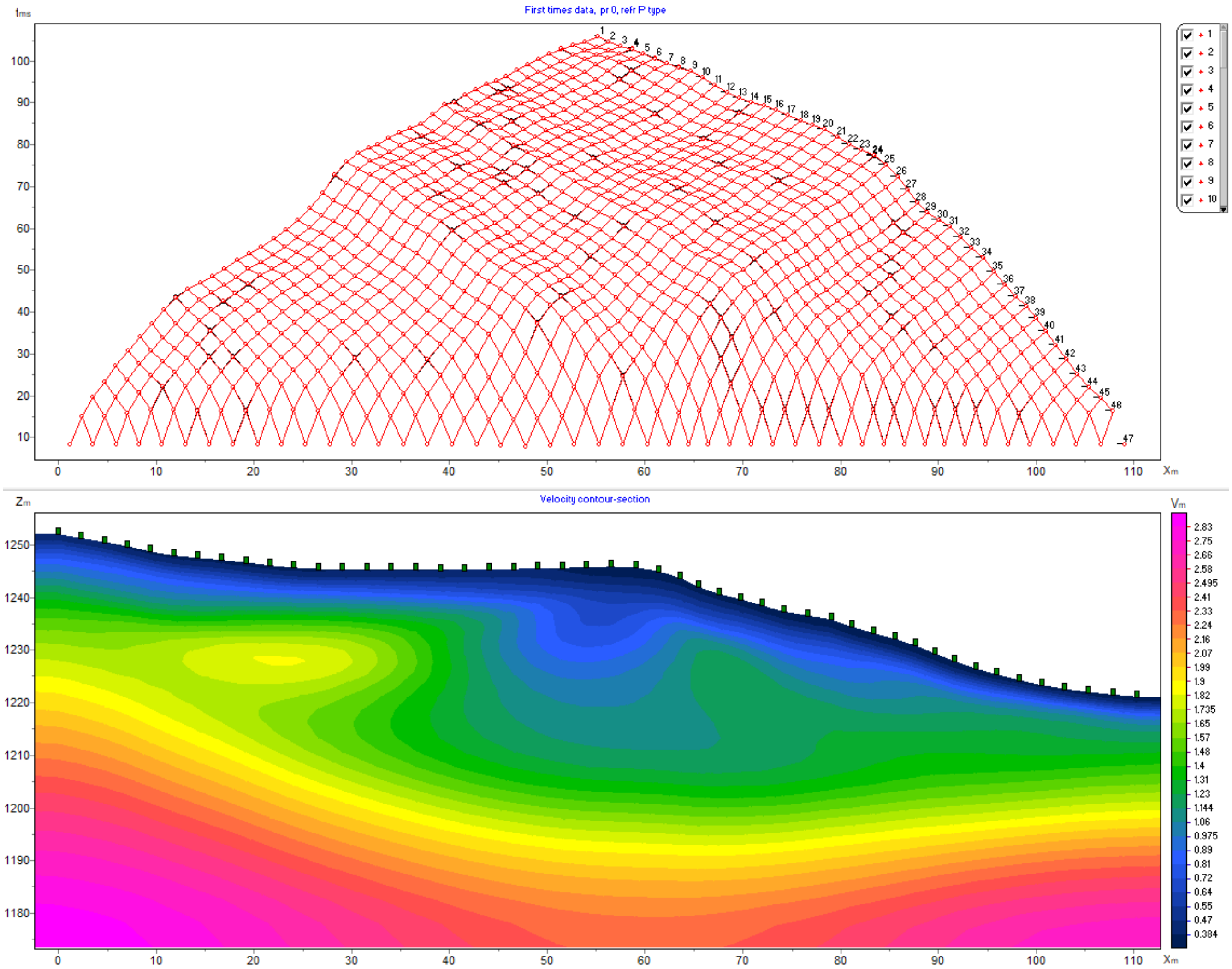
Zond Software



ZondST2D

Program for processing and two-dimensional interpretation of seismic tomography data (land, borehole and marine surveys)

With MASW, amplitude inversion and anisotropy inversion modules

User manual

Table of contents

Introduction

ZondST2D is a program for two-dimensional interpretation of seismic refraction tomography (SRT) data for land, cross-hole and marine surveys. Additionally, the program allows you to display and process seismic records, pick first arrivals, analyze active and passive surface wave data (MASW), perform analysis of amplitudes, velocity anisotropy, and reflected waves.

ZondST2D is an effective tool for automatic and interactive interpretation of SRT data with a user-friendly interface and a variety of visualization options. The program allows you to solve a wide range of problems from raw data processing to mathematical modeling and sensitivity analysis.

The program interface consists of the main module intended for solving the forward and inverse problems of SRT and several additional modules dedicated to specific tasks such as first arrival picking, processing of MASW data, and others.

A special interface was developed for the processing of field records, designed to facilitate and automate the picking process, with the main emphasis on the variety of visualization methods and accessibility of frequently used functions. The user can simultaneously pick two types of refracted waves (P and S) and up to three reflected waves. The program interface is optimized for the joint interpretation of compressional (P) and shear (S) refracted waves.

When solving the forward problem (ray tracing), a special algorithm of graph theory – the *Shortest path method* – is used. The algorithm performs the calculation of the shortest propagation path of the refracted wave. Tracing the shortest path from both the source and receiver to a reflector allows computing the reflected wave path for each boundary. The section of the boundary with the minimum total travel time from source to receiver is selected as a point of reflection. This algorithm is characterized by high computing rates and controlled accuracy.

The forward problem (i.e. the calculation of seismic ray paths for an arbitrary 2D model) can be solved in **ZondST2D** using three approaches:

1) Ray approximation with constant velocities inside the model cells.

2) Ray approximation with linear velocity changes inside the model cells (the velocities are specified for cell nodes).

3) Modeling of a ray beam of variable radius (ray spreading) with linear velocity changes inside the model cells (the velocities are specified for cell nodes).

Tomographic inversion allows you to obtain seismic velocity cross-sections for both compressional (P) and shear (S) waves. Sources and receivers can be located on the ground surface, in boreholes, and on the bottom or on the surface of a water body. The program can be used for the interpretation of vertical seismic profiling (VSP) data. Along with seismic velocities, the algorithm for determination of velocity anisotropy distribution is implemented. The program uses a simple version of the velocity anisotropy coefficient – the Vx/Vz ratio. Taking anisotropy into account has a particularly strong effect on the results of cross-hole data interpretation.

The layered modeling mode is intended for creating and editing layered models. In this mode, the velocity model is defined by a set of layers with arbitrary geometry and velocity distribution in each layer. A layer boundary can be reflective and refractive, or refractive only. The advantage of this modeling mode is the possibility of joint inversion of P and S-wave data within the same model geometry; it is also used for non-tomographic interpretation of refraction data. The program implements an accurate solution of forward and inverse problems for a layered medium, allowing the wave to propagate along the shortest distance path according to the Fermat principle, contrary to the approach implemented in the reciprocal method where the wave always propagates along a refractive boundary.

The MASW module allows you to obtain shear wave velocity depth profiles through analysis of active (MASW) and passive (ReMi) surface wave data. The MASW field measurements technique is practically the same as that of the refraction method, thus, the Vs sections can be obtained directly from your refraction data. The program implements a full cycle of data processing from obtaining dispersion curves to building velocity cross-sections and supports the inversion of higher modes.

The amplitude inversion module implements the *Attenuation tomography* algorithm and allows you to obtain a distribution of the attenuation parameter (Q) in the medium from the first arrival amplitudes. The problem is solved using the velocity model obtained from the first arrival data. The amplitudes values are picked simultaneously with the first arrival times.

ZondST2D uses a simple data format that allows combining different measurement systems, including topography data and other auxiliary information. Commonly used data formats are also supported.

Velocity model building based on expected worksite geology is an important process preceding fieldwork. Through modeling, you can estimate the method’s resolution and optimize the fieldwork technique for solving a specific geological problem. The tomography data modeling can be performed in mesh (grid) and polygonal modeling modes.

Several methods of solving the inverse tomographic problem are realized in the program, the main of which are smoothing inversion to obtain a smooth velocity distribution in the cross-section, block inversion to obtain a blocky velocity distribution, and focusing inversion to obtain a piecewise smooth velocity distribution.

ZondST2D offers a wide range of options to account for a priori information. Considering the equivalence of inverse problem solutions, the reliability of interpretation results depends on the amount of a priori data used. In the program, you can set weights to specific measurements, set and fix limits of velocity change in individual cells, draw known boundaries, use an a priori model as a reference in the inversion process. It is also possible to import and display results of other methods and borehole data, which contributes to a more comprehensive approach to data interpretation. The program performs joint inversion with data obtained with other geophysical methods, allowing you to effectively combine these data with the velocity cross-section on the basis of common framework, geometry of boundaries, or minimization of cross-gradient operator.

Newton's method with regularization is used to solve the inverse problem. Regularization results in a more stable solution and allows you to obtain a smoother velocity distribution in the medium:

$$(A^T W^T \mathrm{WA} + \mu C^T \mathrm{RC})\Delta m = A^T W^T \Delta f - \mu C^T \mathrm{RCm},$$

where A – matrix of partial derivatives of measured values (Jacobian matrix), C – smoothing operator, W – matrix of relative measurement errors, m – vector of cross-section parameters, μ – regularizing parameter, Δf – vector of discrepancies between calculated and measured values, R – focusing operator.

ZondST2D has a powerful set of data visualization tools, a data editor and tools for analysis of sensitivity and resolution of the method.

The program can be used to determine an optimal survey design at the survey planning stage. For this purpose, the software allows you to analyze the resolution of a particular survey array. The essence of the analysis is to let you study the influence of a particular portion of the model on the measurement results using the sensitivity function:

$$S = \sqrt{\mathrm{diag}(A^T A)}.$$

The sensitivity study allows you to make an optimal choice of measurement system for a specific geological problem.

Installing the software

ZondST2D is supplied via the Internet. You can download the latest program updates from our website: www.zond-geo.com.

To install the program, copy the program file to the desired location on your hard drive (e.g., create C:\Zond folder). To install an update, simply overwrite the old version of the file with the new one.

If the **ZondST2D** dongle driver is not installed, you must install the SenseLock dongle driver before starting the program. To do this, open the **SenseLock** folder and run the **InstWiz3.exe** file. The latest version of the driver can be downloaded from the SenseLock website (http://www.senselock.com). After installing the driver, insert the dongle. If the driver was installed successfully, a message will appear in the notification area that the key has been found.

To uninstall the program, delete the folder containing the program.

System requirements

The **ZondST2D** program can be installed on a computer with Windows 98 or higher operating system. Recommended system parameters: Pentium IV 2 Hz CPU, 1 GB RAM, 1024 x 768 screen resolution, 24-bit (True color) color mode (the screen resolution should not be changed when working with data).

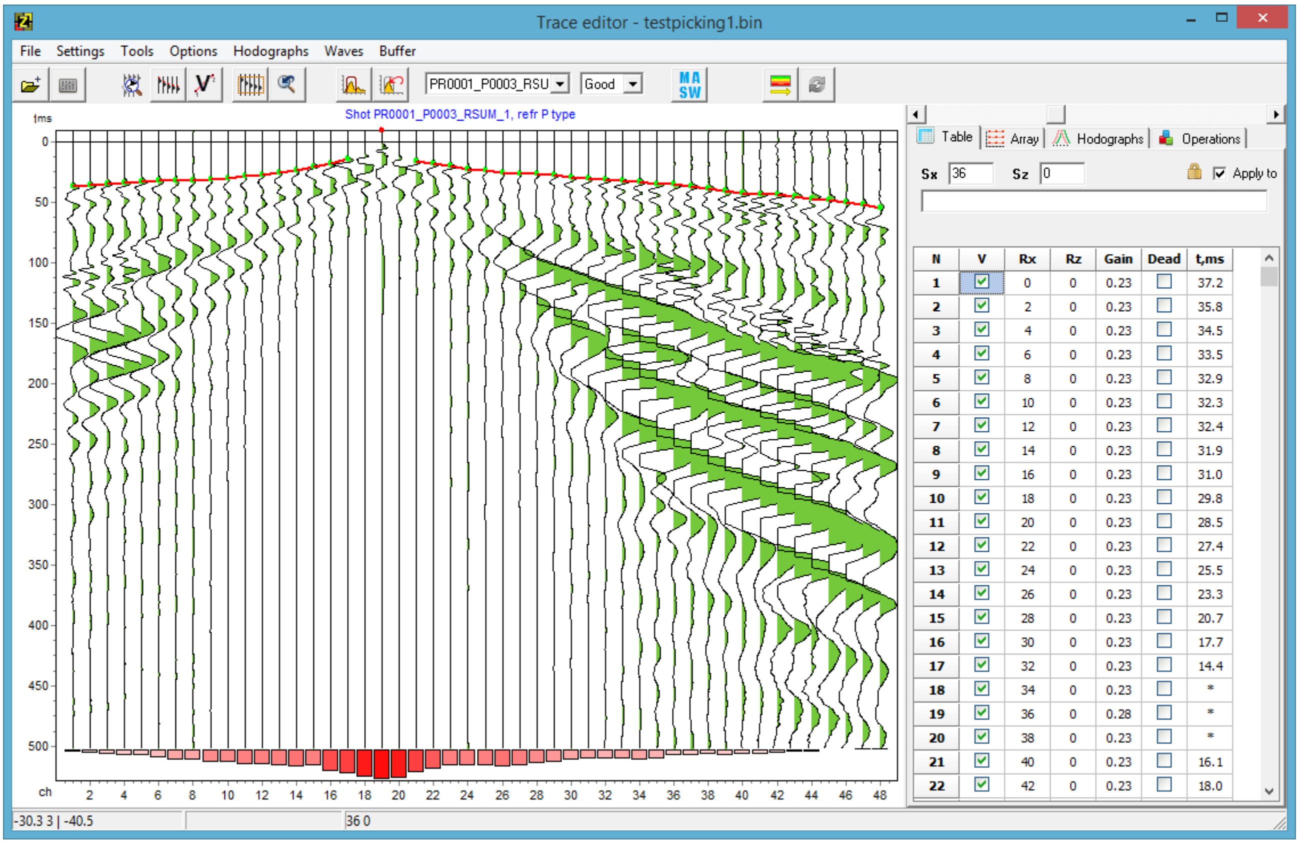
Because of the use of system registry, on operating systems higher than Windows XP the program needs to be run as an administrator (right-click on the program icon – **Run as administrator**).

Units of measurement

In the program, the distance units (coordinates of sources and receivers, depth, topography) are **meters** (m), the time units are **milliseconds** (ms), the velocity units are **kilometers per second** (km/s).

Field data processing in the Trace Editor module

The main purpose of field data processing is to identify and pick the arrival times (or amplitudes) of the target waves. This is carried out in the **Trace Editor** module which can be opened by pressing the  button on the toolbar or using the **Options / Modules / Data picker** menu command. After opening the module you can read in the field data in the SEG-Y or SEG-2 format, assign the geometry and start the picking process.



The module workspace is divided into two sections: the graphic display of the seismic data on the left and the multi-tab section for displaying and changing parameters of the measurement system and picked travel-time curves on the right.

Getting started with the module

Working with the module starts with opening a file or several files in the SEG-Y or SEG-2 format, or a project file. After opening a file, a dialog box appears prompting you to specify whether you want to show detailed information for each file. If you select **Yes**, an editable table will appear in a new window where you can specify the geometry of the seismic record and select traces to be displayed (which also can be done at any time later).

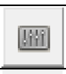
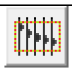
Traces sample.sgy

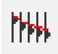
tr	used	shot	tx	rx	tz	rz
1	☑	1	0	40	0	0
2	☑	1	0	45	0	0
3	☑	1	0	50	0	0
4	☑	1	0	55	0	0
5	☑	1	0	60	0	0
6	☑	1	0	65	0	0
7	☑	1	0	70	0	0
8	☑	1	0	75	0	0
9	☑	1	0	80	0	0
10	☑	1	0	85	0	0
11	☑	1	0	90	0	0
12	☑	1	0	95	0	0
13	☑	1	0	100	0	0
14	☑	1	0	105	0	0
15	☑	1	0	110	0	0
16	☑	1	0	115	0	0
17	☑	1	0	120	0	0
18	☑	1	0	125	0	0
19	☑	1	0	130	0	0
20	☑	1	0	135	0	0
21	☑	1	0	140	0	0
22	☑	1	0	145	0	0

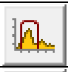
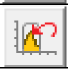
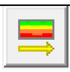
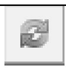
Before reading in data in the SEG-Y format, make sure the SEG-Y file settings are correct (see the **SEG-Y file settings** section for details).

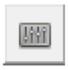
Trace Editor window toolbar

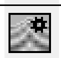
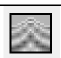
The toolbar is used for quick access to the most frequently used functions. It contains the following buttons:

	Open files in the SEG-Y or SEG-2 format or a project file in the BIN format.
	Bring up the additional menu with frequently used functions.
	Enable the **Zoom** mode. To zoom in on a portion of the seismic record, press the left mouse button and drag the mouse up/down and to the right. To zoom out, drag the mouse in the opposite direction. After activating this mode, the following two buttons appear on the toolbar.
	Set the current zoom setting as the working area. When the working area is set, automatic gain adjustment is applied to the portion of the data in the area.
	Go to the previous zoom setting.
	Enable the picking mode. Use the left mouse button for picking the

	arrivals; right-clicking on a pick point removes the point. The picking can be performed in manual or semi-automatic mode. After activating this mode, the following two buttons appear on the toolbar.
	Enable the *Multipicking* mode. The picking is performed automatically between two specified traces. The picking path is displayed as a line connecting the first specified pick point and the current cursor position. If the *Autopicking* mode is selected, instead of the line, the band in which the search is made and the preliminary picks (in the form of grey circles) are displayed. The width of the search band is adjusted with the mouse wheel with the Ctrl button pressed.
	Enable the *Autopicking* mode for automatic picking of the travel-time curve by one of the specified criteria. In the *Multipicking* mode, the autopicking is performed for all traces in the specified trace range, otherwise, the search is performed for a single trace. It is important to choose correctly the width of the time window (indicated with a circle or band) in which the search for the specified phase is performed. The width of the time window is adjusted using the mouse wheel with the Ctrl button pressed. The search criterion (the signal phase you want to pick) is selected in the pop-up menu brought up by right-clicking on the *Autopicking* toolbar button. Available criteria are: *To null* – search for the transition from null (or close to null) values in the window (the onset picking). *To extremum* – search for the maximum absolute value in the window. *To maximum* – search for the maximum value in the window. *To minimum* – search for the minimum value in the window. *To best correlated point* – search for the point of best correlation with neighboring traces. *Move to reciprocity point* – if there is an assigned reciprocity point for a pick point, their times will be averaged.
	Enable the linear velocity approximation mode. To determine the velocity, position the cursor on the starting point of the velocity approximation line, press and hold down the left mouse button and drag the cursor to the desired position. The velocity value is displayed in the status bar (the

	second section). The array geometry should be specified for correct velocity determination. If a reflected wave is selected in the **Waves** section of the menu, you can determine the average velocity of a reflected wave using NMO approximation (hyperbola fitting). To determine the velocity, press the V key; the hyperbola will appear at the current cursor position. The velocity is adjusted using the mouse wheel and displayed in the status bar. Pressing the right mouse button will build a travel time curve corresponding to the current position of the hyperbola.
	Open the **Filter&Spectrum** dialog box.
	Cancel filtering (restore the unprocessed data).
	Select the active field record from the drop-down list containing all records opened in the project.
	Set the pick quality for the active field record. This estimate will be used in the inversion.
	Open the **MASW/ReMi** module. The survey geometry should be specified beforehand.
	Create a measurement system and initial model and go to the main module of the program.
	Update picks for the current project and go to the main module of the program. The survey geometry should not be changed.

The  toolbar button brings up the additional menu containing the following options:

	Open the dialog box for configuring SEG-Y reading options.
	Open the dialog box for changing graphic settings of field records and travel-time curves.
	Open the dialog box for changing the scale and muting of field records.
	Open the dialog box for changing graphic settings of the variable density display mode.
	Enable the active trace selection mode. The active trace is highlighted with a different color. This mode should not be used during the picking as it slows down the program performance.
	Enable the variable density display of the field record in the

	background.
	Rotate the field record by ninety degrees. Depending on the data type, you can choose the convenient orientation of the field record.
	Show the power profile of the seismic record. The power characterizes the energy of the seismic signal, which varies depending on the distance from the source.

Description of the Trace Editor menu commands

The table below provides the description of the **Trace editor** module menu commands. Some of the commands duplicate the toolbar buttons.

File	Open SEG-Y/Project	Open files in the SEG-Y or SEG-2 format or a project file in the BIN format.
File	Add to project	Add data to the project. The option is used to combine the data collected at different times into one project.
File	Save project	Save the data, processing results, picks and module settings as a BIN project file.
File	Close project	Remove all data from the current project.
File	Print preview	Open the **Print preview** dialog box.
File	Import topography	Read in a file with topography data. Two-column text files are used to import the topography.
File	Run MASW module	Run the MASW module for the surface waves analysis of the data loaded into the project.
File	Run MASW for group	Run the MASW module for the surface waves analysis of the data loaded into the project. The data should consist of several shot records obtained for the same position of the spread at different shot locations. In some cases, combining several shot records improves the quality of the resulting dispersion image.
Settings	Survey type	Open the dialog box for setting the parameters of the measurement system. It is important to specify the type of survey before starting the picking process, particularly for the cross-hole surveys.

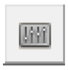
Settings	Change orientation	Rotate the field record by ninety degrees. Depending on the data type, you can choose the convenient orientation of the field record.
Settings	Show background	Enable the variable density display of the field record in the background.
Settings	Background settings	Open the dialog box for changing graphic settings of the variable density display mode.
Settings	Plot's color	Open the dialog box for changing the background color of the graphic area.
Settings	Trace/hodograph settings	Open the dialog box for changing graphic settings of field records and travel time curves.
Settings	Scale/muting settings	Open the dialog box with records' scaling and muting options.
Settings	Amplitudes collect settings	Open the dialog box for setting the amplitude collecting parameters.
Settings	Working area / Display rectangle	Show the working area frame.
Settings	Working area / Set working area	Set the current zoom setting as a working area. When the working area has been set, automatic gain adjustment is applied to the portion of the data in the area. If the **Apply for next shots** option is activated in the **Table** tab of the multi-tab section, the working area is set for all consequent records opened in the project.
Settings	Working area / Select all	Set the entire field record as the working area.
Settings	Units	Specify the units displayed on the graphic section axes: channel number, X coordinate or offset from the source for the distance axis; time or samples for the time axis.
Settings	Set sample time	Set the data sampling rate in milliseconds (if not specified or specified incorrectly in the data file).
Tools	Filtering	Open the **Filter&Spectrum** dialog box.
Tools	Undo filtering	Cancel filtering (restore the unprocessed data).
Tools	Picking mode	Enable the picking mode. Use the left mouse button for

		picking the arrivals; right-clicking on a pick will remove it.
Tools	Zoom mode	Enable the **Zoom** mode. To zoom in on a portion of the seismic record, press the left mouse button and drag the mouse up/down and to the right. To zoom out, drag the mouse in the opposite direction.
Tools	Display cross	Show the crosshair for more accurate picking.
Tools	Edit source pos	Open the dialog box for editing source coordinates.
Tools	Set survey line	If the survey line has a complex 2D geometry or the shot locations are non-collinear with the spread, this module allows you to assign a line for the survey profile and project the coordinates of sources and receivers on this line.
Options	Delete current shot	Remove the current field record from the project.
Options	Delete empty shots	Remove the field records for which no picking was carried out from the project (e.g., repeated shot records or noisy data).
Options	Delete dead traces	Delete dead traces. Typically, these traces correspond to faulty or disconnected receivers. The dead traces are selected in the **Table** tab.
Options	Sort shots by filename	Sort loaded seismic records by file name.
Options	Sort shots by position	Sort loaded seismic records by shot location.
Options	Sort all traces	Sort traces of loaded seismic records by receiver coordinates.
Options	Combine shots	Add shot gathers with the same source coordinate to the current record. The option allows you to combine several field records obtained with the same shot location but different positions of the receiver spread (e.g., when the spread moves along the line). The survey geometry should be specified beforehand.
Options	Apply AGC	Apply AGC (Automatic Gain Control) which performs normalization of amplitudes by their average value within a predefined time window. The AGC window is specified in samples and is selected based on the purpose of the gain adjustment (often empirically).
Options	Remove aver in	Gain adjustment based on subtraction of an amplitude value

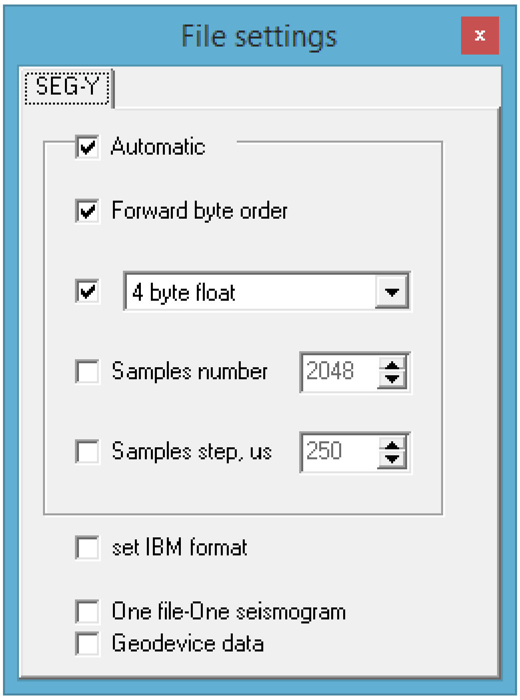
	window	averaged in the specified time window. This option allows you to emphasize the higher frequency signals. The window is specified in samples and is typically selected empirically.
Options	Collect amplitudes from shot	Collect the signal amplitudes at the pick point positions for the current seismic record. The option allows you to collect the amplitudes not at the exact pick point position, since it may be zero in the case of picking the onsets, but in some window around it. The window (in samples) and the collection method are specified in the **Collect settings** dialog box (**Settings / Amplitudes collect settings**).
Options	Collect amplitudes from all	Collect the signal amplitudes at the pick point positions for all seismic records.
Options	Undo action	Undo the last picking action performed.
Hodographs	Delete current	Delete the current pick.
Hodographs	Autocorrect current	Automatic correction of the current pick (the already picked arrivals are correlated based on the shape of the signal near the pick points).
Hodographs	Project current	Automatic picking mode based on the projection of the neighboring seismic records' picks onto the current record. The resulting pick can be used as a draft for the subsequent manual picking.
Hodographs	Project empty times	Automatic picking mode based on the projection of the neighboring seismic records' picks onto the current record. The automatic picking is performed only for those traces that have not been picked already (e.g., the traces are noisy).
Hodographs	Copy current	Copy the current pick to the clipboard.
Hodographs	Paste to current	Paste the pick from the clipboard to the current record.
Hodographs	Load picks	Read in picks from a file in one of the below formats.
Hodographs	Save picks	Save a pick file. The following options are available: *Laccolite god file* – save the current pick to a GOD file; *Laccolite directory* – save all picks to GOD files in a specified directory; *SRT file* – save picks to an SRT file.
Hodographs	Smooth all	Smooth all picks (a special algorithm that takes into account the reciprocity principle is used for smoothing).

Hodographs	Smooth current	Smooth the current pick (a special algorithm that takes into account the reciprocity principle is used for smoothing).
Hodographs	Correct start times (reciprocity)	Correct start times of shot records based on reciprocal times of the corresponding picks.
Hodographs	Average reciprocity data	Average reciprocal times of picks.
Hodographs	Correct start times (calculated)	Correct start times of shot records based on theoretical (calculated) times of travel time curves obtained through 2D inversion.
Hodographs	Calc reciprocity error	Show the average error of data correlation calculated based on the reciprocity principle.
Hodographs	Xpos=(S+R)/2	If this option is enabled, the travel time curve X coordinate is calculated as a midpoint between the source X and receiver X; if disabled, the receiver coordinates are used (only applies to the travel time curves plotted in the **Hodographs** tab).
Waves	P-refracted	Specify the picked arrivals as compressional (P) refracted wave. Only the picks of the active (current) wave type are displayed.
Waves	S-refracted	Specify the picked arrivals as shear (S) refracted wave.
Waves	Reflected-1	Specify the picked arrivals as reflected wave (first contact).
Waves	Reflected-2	Specify the picked arrivals as reflected wave (second contact).
Waves	Reflected-3	Specify the picked arrivals as reflected wave (third contact).
Waves	Exchange P&S	Change the wave type from P-refracted to S-refracted and vice versa. This option can be used if the wrong wave type was specified during picking.
Waves	Remove active data	Delete all picks for the current wave type.
Buffer	Picks 1-5	The buffer allows you to store up to five picks. By selecting one of the buffer picks you can save/load the current pick into/from the buffer. The buffer slots that contain stored picks are indicated by checkmarks. If the selected buffer slot is not empty (checked), in the dialog box that appears you can choose to load the buffer pick (**From Buffer**) or to save a current pick to this buffer slot (**To buffer**).

SEG-Y file settings

The **File Settings** dialog box for changing specifications of SEG-Y files is accessible through the pop-up menu brought up by pressing the  toolbar button.

The dialog box contains options controlling the reading of field data. SEG-Y is a standard format for storing seismic data, however, some seismographs may deviate from the standard form, and in this case, it is necessary to select the recording parameters to read in the data correctly before opening a SEG-Y file.



Automatic – enables the automatic format detection mode (works in most cases). If this option is disabled, the following settings become available.

Forward byte order – sets the byte reading order. The drop-down list below allows selecting the data type.

Samples number – sets the number of samples in a trace.

Samples step, us – sets the sampling interval (in microseconds).

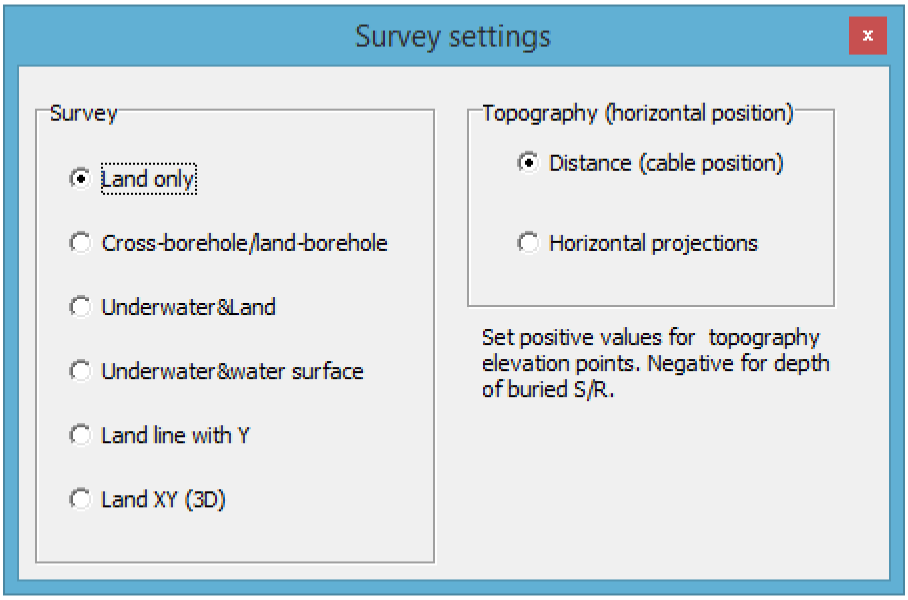
Set IBM format – sets the IBM floating point number format.

One file-One seismogram – specifies that each opened file contains one shot gather.

Typically, only the **Set IBM format** and **Forward byte order** options need to be changed.

Setting the survey type

The dialog box for selecting a survey type is opened using the **Settings / Survey type** command. The survey type should be specified before starting the picking process.



The **Survey** group box contains the following survey type options:

Land only – for land surveys, the table in the **Table** tab contains topography data in the **Rz** column. The elevation/depth of the source can be specified in the **Sz** input box (negative values should be entered for depths below the ground surface).

Cross-borehole/land-borehole – for cross-hole and downhole surveys, the table in the **Table** tab contains borehole depths in the **Rz** column; the depth of the source/receiver can be specified in the **Sz**/**Rz** input box (0 if the source/receiver is located on the ground surface).

Underwater&Land is a combined survey consisting of marine (bottom) and land portions.

Underwater&water surface is a marine survey that can consist of bottom and water surface portions.

Land line with Y – 2D survey along a curved line or with shot locations non-collinear with the spread. When this type is selected, the Y coordinates can be specified (Y is the direction perpendicular to the direction of the survey line).

Land XY (3D) – can be used when a portion of a 3D survey is used for 2D interpretation. When this type is selected, the Y coordinates can be specified (Y is the direction perpendicular to the direction of the survey line).

The **Topography (horizontal position)** group box contains options determining the type of X coordinate:

Distance (cable position) – the survey geometry is specified in distances along the line (cable).

Horizontal projection – the survey geometry is specified in horizontal projections (real coordinates, with the account of topography).

Changing the scale of field records

Interactive scaling of field records can be performed in the **Zoom** mode enabled by pressing the  toolbar button. To zoom in on a portion of the seismic record, press the left mouse button and drag the mouse up/down and to the right. To zoom out to the original scale, drag the mouse in the opposite direction. The  button returns the view to the previous zoom setting.

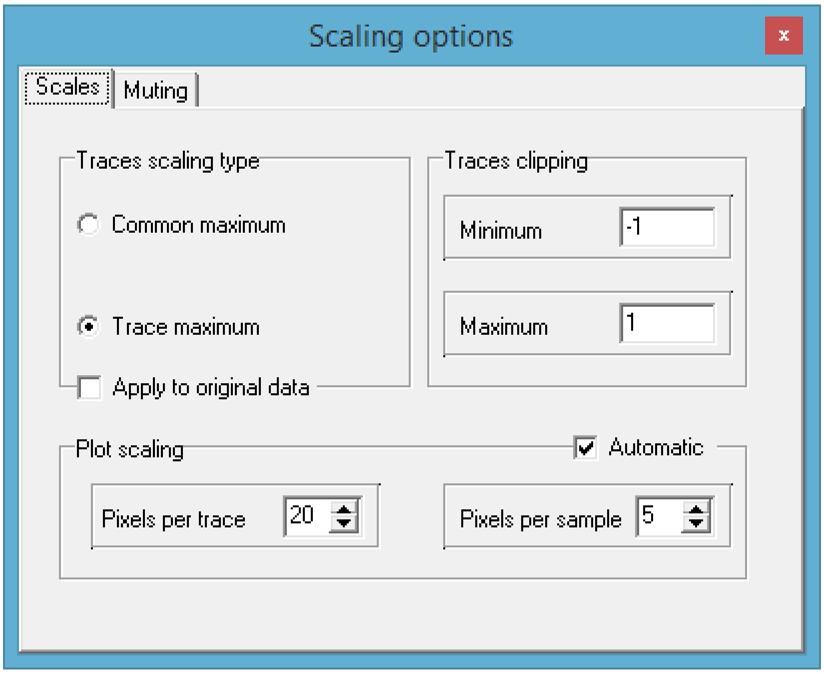
The field record can be moved in the graphic area in all operating modes by dragging it with the right mouse button pressed.

Additional scaling options are available in the **Scaling options** dialog box described in the following section.

Trace normalization and muting

The **Scaling options** dialog box containing field record scale, gain and muting settings is opened using the **Settings / Scale/muting settings** command. The dialog box is also accessible through the pop-up menu brought up by pressing the  toolbar button.

The dialog box contains two tabs. The **Scales** tab contains plot scaling, trace normalization and trace clipping options.



The **Traces scaling type** group box defines the trace normalization type.

Common maximum – normalization to the maximum value of the entire record.

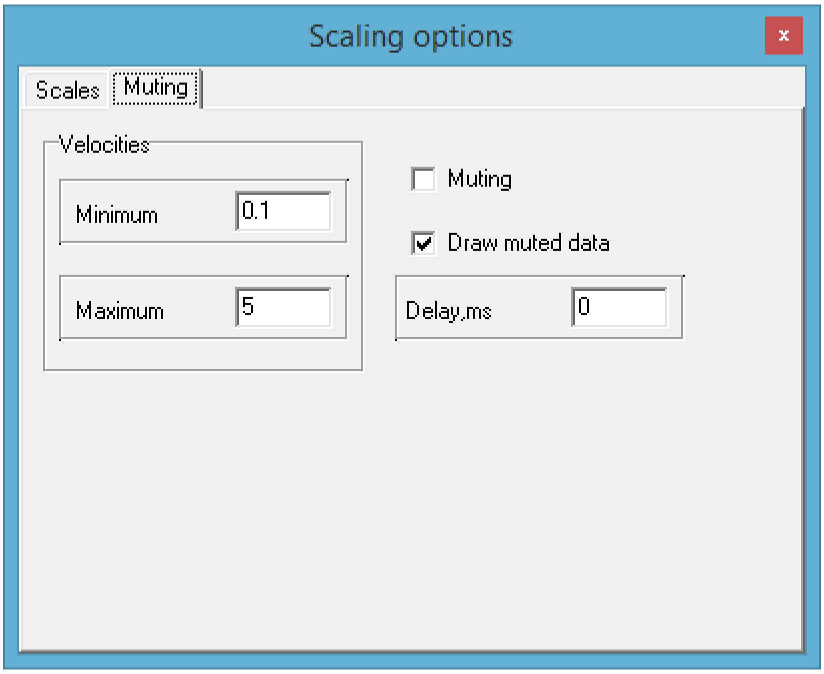
Trace maximum – normalization to the maximum value of each trace.

If the **Apply to original data** checkbox is activated, the normalization is performed using the values of the original (unprocessed) data.

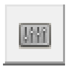
Using input boxes in the **Traces clipping** group box, you can specify the minimum and maximum trace clipping values, assuming the distance between neighboring traces equals 1.

When the **Automatic** checkbox in the **Plot scaling** group box is activated, the distance and time scales are selected automatically to display the field record in full scale. When disabled, the scales are specified using the **Pixel per trace** and **Pixel per sample** input boxes.

The **Muting** tab contains options that control the muting of field records. The muting is performed based on the set velocity limits in the **Velocities** group box. The **Muting** option turns on the display of the set velocity range boundaries on the record. If the **Draw muted data** checkbox is unchecked, the data outside the specified velocity range is not displayed. Using the **Delay, ms** input box, you can set a time delay applied to the lower boundary of the velocity range.

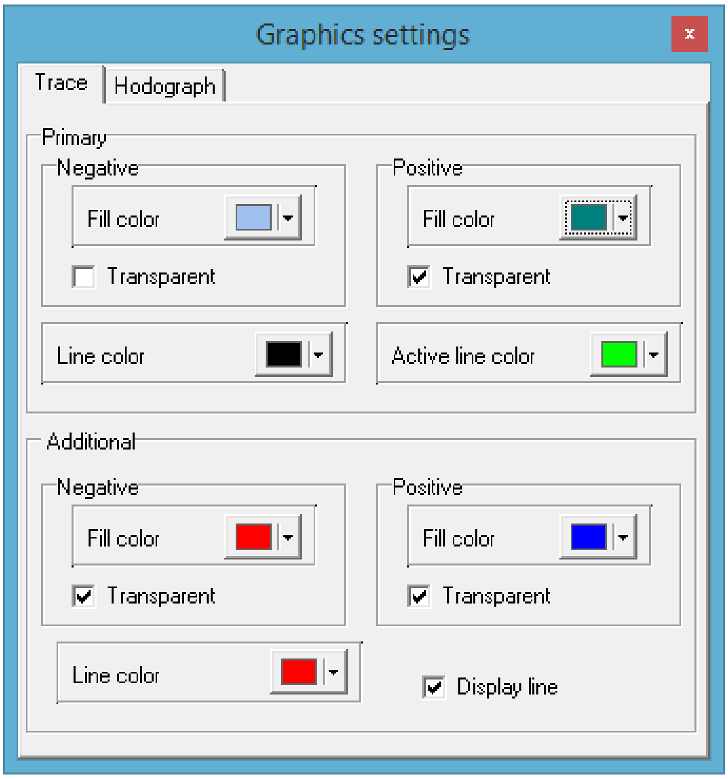


Graphic settings

The dialog box for changing graphic settings of traces and picks is opened using the **Settings / Trace/hodographs settings** command. The dialog box is also accessible through the pop-up menu brought up by pressing the  toolbar button.

The **Trace** tab of the dialog box contains the settings defining the appearance of traces of seismic records. It is divided into two group boxes – **Primary** and **Additional**, containing the settings for the main and additional (displayed over the main one in the *Multi* mode) records, respectively.

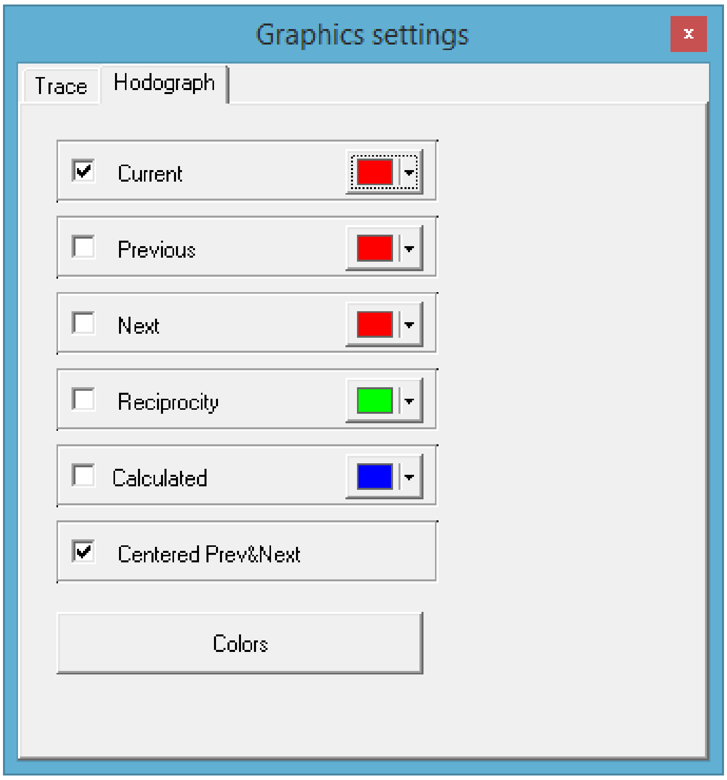
The **Negative** and **Positive** boxes contain the fill color and transparency settings for negative and positive amplitudes, respectively.



Line color – sets the color of wiggle traces.

Active line color – sets the color of the active trace line.

Display line – show/hide wiggle traces.



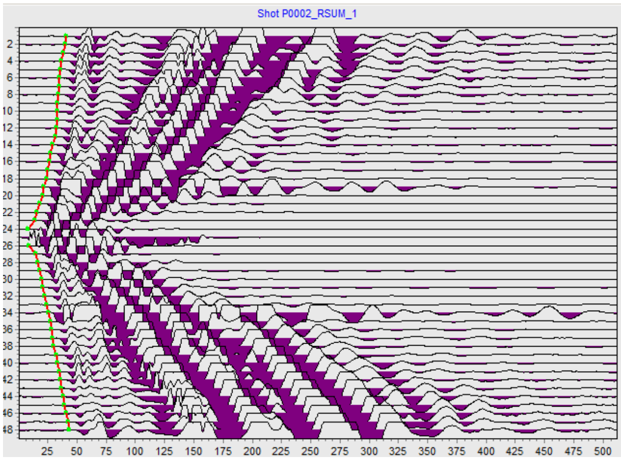
The **Hodograph** tab contains the color settings of picks displayed on the seismic records. You can change the color and enable/disable the display of the current, previous, next, reciprocal and calculated picks.

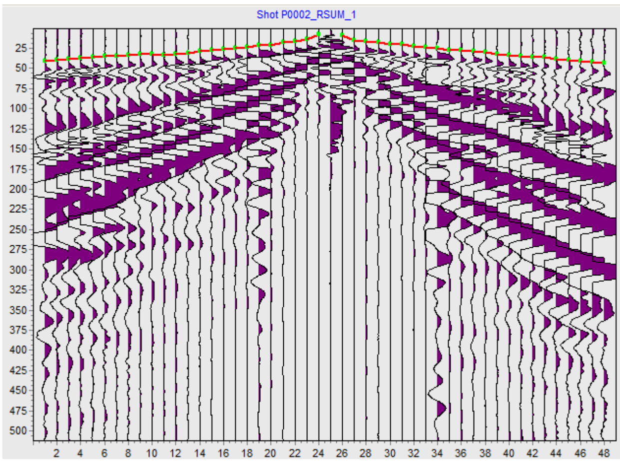
Centered Prev&Next – if this option is activated, the previous and next picks are plotted using offsets from shot locations instead of receiver coordinates.

The **Colors** button opens the dialog box for changing graphic settings of the travel time curves displayed in the **Hodographs** tab of the module.

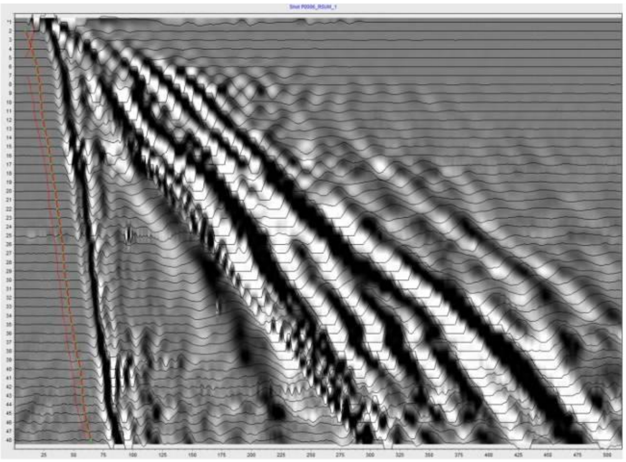
View options

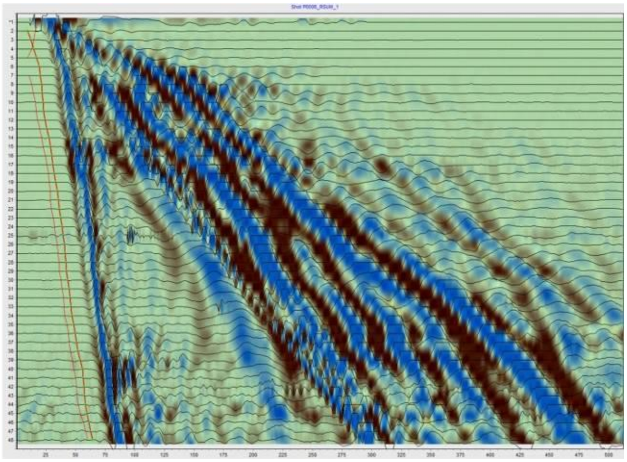
Depending on the data type and personal preferences, it may be convenient to use vertical or horizontal orientation of seismic records. The **Settings / Change orientation** command allows you to rotate the record by 90 degrees.





To enable the variable density display of seismic records, select the **Settings / Show background** menu command. Colors can be adjusted in the **Color palette** dialog box opened using the **Settings / Background settings** command. These commands are also duplicated in the pop-up menu brought up by pressing the toolbar button. By default, the variable density display is shown on the background; to display only the variable density, set the transparency for wiggle traces and color fill in the trace graphic settings dialog box.



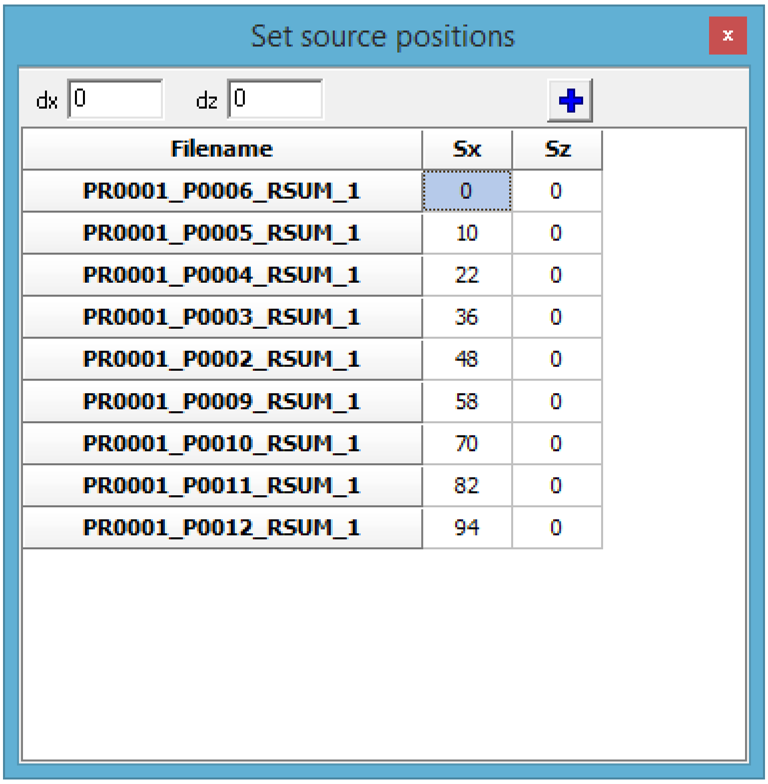


Geometry input

If data files do not contain geometry, you can input it in the module. Some of the options allowing to quickly set the survey geometry are also described in the **Multi-tab section of the module** section below.

If your data contains repeated measurements, it is convenient to enter the source coordinates first, then sort the records by shot location. This will allow you to look through the data and delete unneeded records, if necessary.

The source coordinates for a list of field records can be entered using the **Set source positions** dialogue box opened with the **Tools / Edit source pos** command.



Filename	Sx	Sz
PR0001_P0006_RSUM_1	0	0
PR0001_P0005_RSUM_1	10	0
PR0001_P0004_RSUM_1	22	0
PR0001_P0003_RSUM_1	36	0
PR0001_P0002_RSUM_1	48	0
PR0001_P0009_RSUM_1	58	0
PR0001_P0010_RSUM_1	70	0
PR0001_P0011_RSUM_1	82	0
PR0001_P0012_RSUM_1	94	0

The coordinates can be edited directly in the table or assigned using the following procedures.

The **dx** and **dz** input boxes are used to specify the X (for land surveys) and Z (for borehole surveys) coordinate increments. To apply the increment, select a file (a record) in the list and press the  button. The coordinate of the next file will change to the X/Z value of the selected file plus the increment value. To assign the same coordinate to a range of files, first, enter the coordinate value for a file located at one end of the range, then select the file located at the opposite end of the range and left-click on the entered value while holding down the Shift key. To assign incremented coordinates to a range of files, first enter a coordinate value for a file located at the top of the range (starting coordinate), then right-click on this coordinate; after entering the increment value in the dialog box that appears and pressing OK, the coordinates of the following files will be incremented.

In the **Table** tab of the module, the receiver coordinates can be edited either by entering them one by one or using the procedure for assigning incremented coordinates described above. If the **Apply to next shots** option is activated, the entered receiver coordinates will be copied to all subsequent seismic records (but not to previous records).

Working with the module

Loaded shot records can be sorted:

- by filename – **Options / Sort shots by filename**;

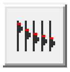
- by shot location – **Options / Sort shots by position.**

The amplitude gain of a field record can be changed by left-clicking (to increase) and right-clicking (to decrease) on the **Gain** column header in the **Table** tab of the multi-tab section. The amplitude gain of an individual trace can be changed using the mouse wheel while hovering over the trace in the graphic display section.

Stacking and subtraction of gathers can be performed using options in the **Operations** tab of the multi-tab section (see the **Multi-tab section of the module** section for details).

To correct shot (trigger) times of field records, the 

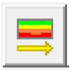
box in the **Hodographs** tab is used. Positive input values are used to correct for a delayed trigger, negative values – to correct for a pre-shot trigger. To apply the time correction to all seismic records, press the flag button.

The picking mode is enabled by pressing the  toolbar button. Use the left mouse button for picking the arrivals; right-clicking on a pick point removes the point. By default, the picks are displayed in red, the travel time curves calculated in the result of inversion – in blue.

Reciprocal times of first arrival picks can be displayed in the form of points (or circles) during picking. For this, the **Reciprocity** checkbox should be checked in the **Hodograph** tab of the trace/hodograph settings dialog box (**Settings / Trace/hodographs settings**).

To switch to a reciprocal shot record (if present), press and hold Ctrl+Alt and left-click on the trace corresponding to the shot location of the reciprocal record.

An action (picking, coordinate input) can be canceled with the Ctrl+Z key combination.

After picking is completed, press the  toolbar button to create a measurement system and initial model and go to the main (inversion) module of the program.

Extraction of first arrival amplitudes

The dialog box for specifying the amplitude collection settings is opened using the **Settings / Amplitudes collect settings** command.

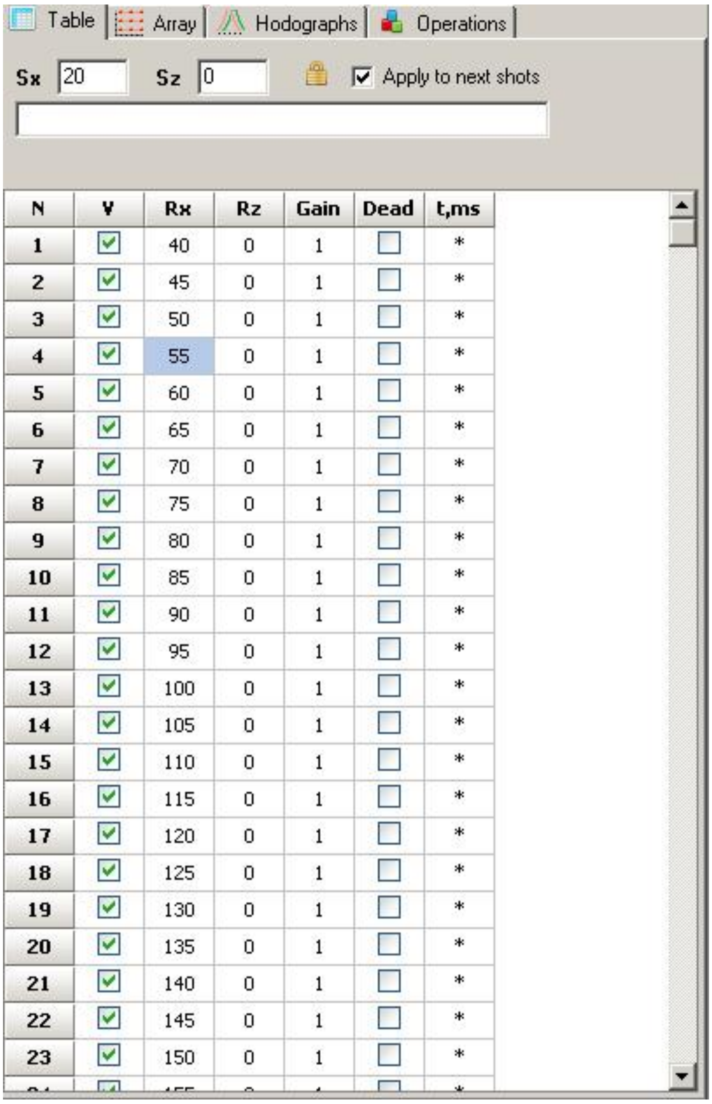
Style – sets the method of amplitude collection: **Around t0** (if picking of signal maximum was performed) or **After t0** (if picking of signal onset was performed).

The **Window** input box specifies the size of the window (in samples) where the search for the maximum amplitude will be performed. The window size should be chosen according to the signal frequency (approximately half the period).

Multi-tab section of the module

The multi-tab section of the **Trace editor** module consists of several tabs.

The **Table** tab contains a table with coordinates of receivers, gain values and other settings for the current field record.



N	V	Rx	Rz	Gain	Dead	t,ms
1	☑	40	0	1	☐	*
2	☑	45	0	1	☐	*
3	☑	50	0	1	☐	*
4	☑	55	0	1	☐	*
5	☑	60	0	1	☐	*
6	☑	65	0	1	☐	*
7	☑	70	0	1	☐	*
8	☑	75	0	1	☐	*
9	☑	80	0	1	☐	*
10	☑	85	0	1	☐	*
11	☑	90	0	1	☐	*
12	☑	95	0	1	☐	*
13	☑	100	0	1	☐	*
14	☑	105	0	1	☐	*
15	☑	110	0	1	☐	*
16	☑	115	0	1	☐	*
17	☑	120	0	1	☐	*
18	☑	125	0	1	☐	*
19	☑	130	0	1	☐	*
20	☑	135	0	1	☐	*
21	☑	140	0	1	☐	*
22	☑	145	0	1	☐	*
23	☑	150	0	1	☐	*

The **Sx** and **Sz** (**Sy**) input boxes set the horizontal and vertical source coordinates of the current field record. These coordinates can also be set in the **Set source positions** dialog box described earlier.

Right-click on the **Sx** input box allows you to set the X coordinate increment for all subsequent field records.

The input field below is used for entering a comment for the current record.

If the **Apply to next shots** checkbox is checked, all changes made in the **Rx** (**Ry**), **Rz**, **Gain** and **Dead** columns of the table are applied to all subsequent field records.

The checkboxes in the **V** column enable/disable seismic traces. Left-click on the column header enables all traces, right-click – disables all traces.

The **Rx** (**Ry**) column contains the horizontal coordinates of receivers. Right-click on a table cell allows you to set a coordinate increment for all subsequent cells.

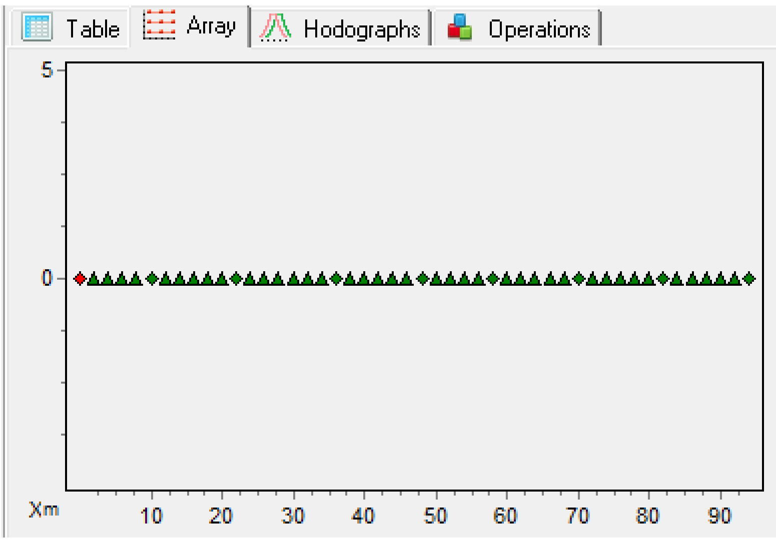
The **Rz** column contains the vertical coordinates of receivers (in the case of borehole surveys – the depth). Right-click on a table cell allows you to set a coordinate increment for all subsequent cells.

The **Gain** column contains the amplitude gain values. Right-clicking on a cell allows you to set a gain value to all subsequent cells (right-click on the top cell to set a gain value to all traces). Left-clicking on the **Gain** header will increase the gains, right-clicking – decrease. To change the polarity of the traces, enter negative values. Double-clicking on a cell will switch the polarity of the corresponding trace.

The checkboxes in the **Dead** column indicate dead traces. Picking for dead traces is not available.

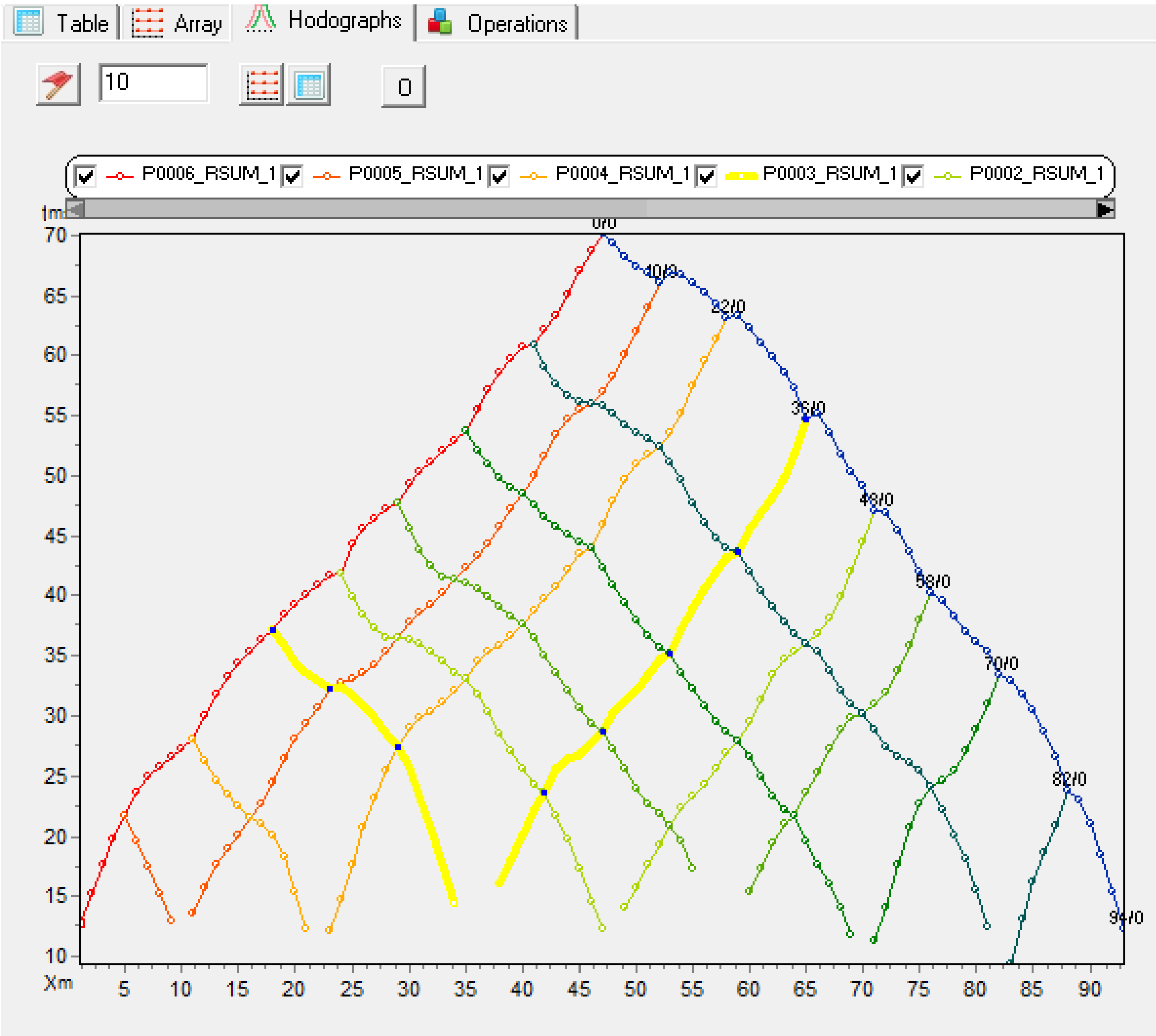
The **t,ms** column contains pick times (in milliseconds or samples depending on the selected measurement unit).

The **Array** tab contains a graphic representation of the survey setup (the survey geometry should be specified for correct rendering).



The source location for the current shot record is displayed in red. Left-clicking on another shot point while holding down the Ctrl key allows you to switch to the corresponding shot record.

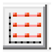
The **Hodographs** tab is intended for displaying and editing the travel time curves. During the picking process, the travel time curves are plotted in this tab. Left-clicking on a curve allows you to switch to the corresponding shot record.



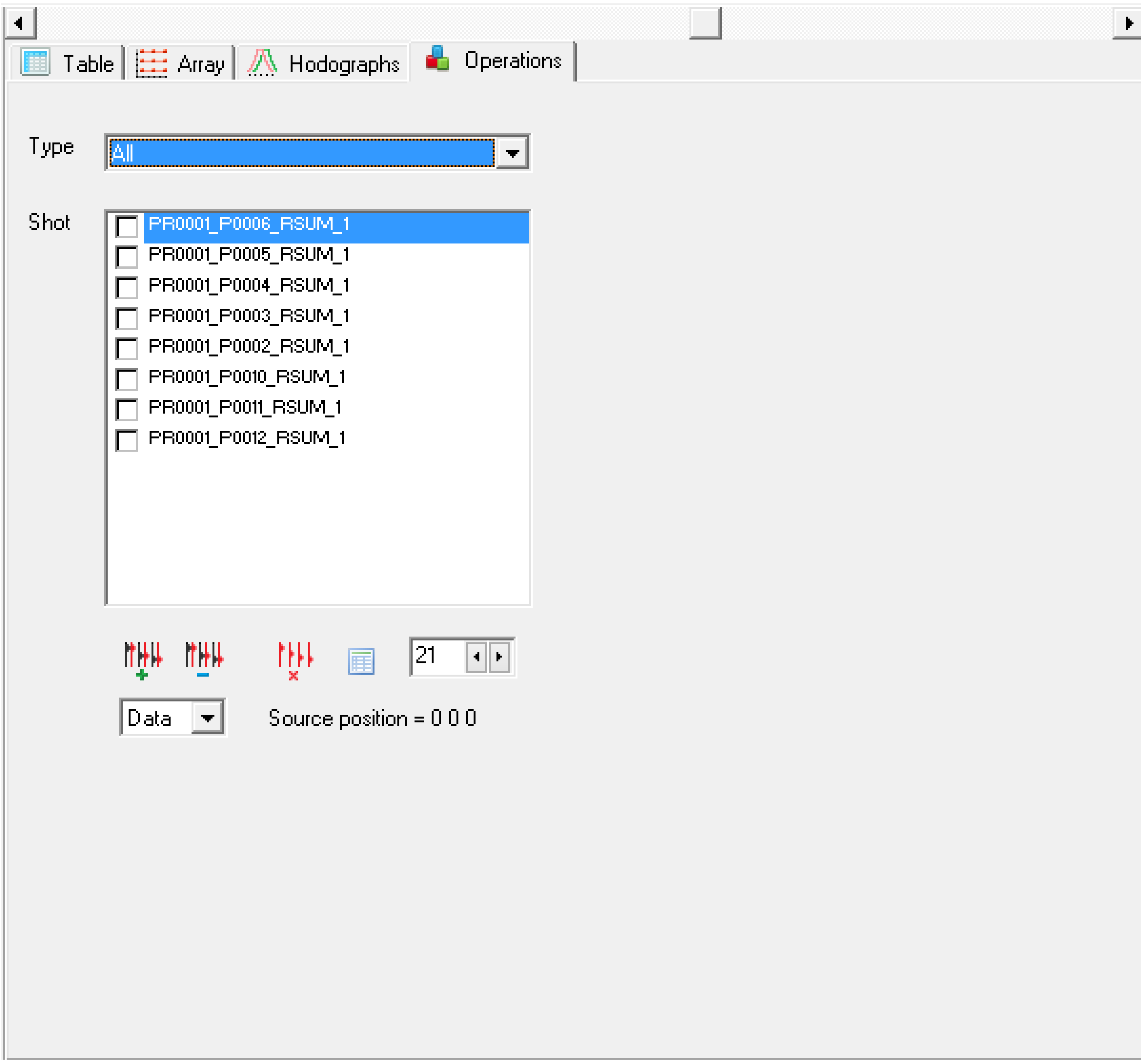
The picked values can be edited by dragging the graph points with the mouse. The entire travel time curve can be moved in the time direction by dragging a graph point while holding down the Shift key; the pick in the seismic display area is shifted correspondingly. Moving a travel time curve by dragging a graph point while holding down the Ctrl key does not shift the pick in the seismic display area but changes the shot (trigger) time in the input box at the top of the tab; this value can then be used to correct the shot time of the record (see below).

To show only one curve, left-click on the curve's ID in the legend while holding down the Shift key; click again to reverse this action. If there is a large number of curves, the mouse wheel can be used to scroll through them. For this, select several adjacent curves in the legend and spin the mouse wheel, with the mouse pointer hovering over the legend.

The  box is used for correction of shot (trigger) times of field records. Positive input values are used to correct for a delayed trigger, negative values – to correct for a pre-shot trigger. To apply the time correction to all seismic records, press the flag button.

The  and  buttons open the **Table** and **Array** tabs in separate windows, which facilitates working with data, allowing you to see the complete set of information for the filed record being processed.

The  button brings up a pop-up menu for selecting the type of data displayed. You can select arrival times, apparent velocities or amplitudes.



The **Operations** tab provides the option to display two gathers at a time for the purpose of comparing, stacking and subtracting the gathers. The secondary gather is displayed in the form of wiggle traces overlaying the current gather. The geometry of the seismic records should be specified beforehand.

The **Type** drop-down list defines the sorting method of available secondary gathers. The secondary gather to display is selected from the list of gathers beneath.

One – only the current gather is displayed.

Source – shows shot gathers with the same shot location and receiver geometry as the current one.

All – shows all available gathers including those with different geometry.

Reciprocity – shows reciprocal gathers.

The operations that can be performed on the gathers are described in the next section.

Stacking and subtraction of gathers

If several shot gathers are obtained at the same shot location, they can be stacked (in the case of repeated measurements) or subtracted (in the case of shear wave data when shots of opposite polarity are recorded).

After selecting the secondary gather to be displayed over the current gather, you can perform the following operations (to perform stacking, subtraction and removing of gathers, check the corresponding checkboxes in the list of secondary gathers).

The  box sets the time shift for the secondary gather in samples.

The  button performs stacking of the primary (current) and secondary gather.

The  button subtracts the secondary gather from the current one.

Options in the  box are used for previewing the results before performing an operation. When you select one of the options from the drop-down list, the results of stacking (if **Sum** is selected) or subtracting (if **Sub** is selected) are shown instead of the initial data of the secondary gather.

You can delete the gathers no longer required from the project by selecting (checking) them in the list and pressing the  button (e.g., when you have performed stacking of several gathers collected at the same shot location and no longer need gathers other than the stacked one).

The button calculates the coefficient of correlation between the primary and secondary gathers. Analyzing the coefficients facilitates the selection of gathers to be stacked. The correlation coefficient is displayed to the right of the gather name in the list.

In the **Reciprocity** mode, reciprocal traces are plotted over the traces of the current shot gather. This allows you to evaluate the quality of the field data, detect trigger time errors and control the accuracy of picking.

Picking

First arrival picking in **ZondST2D** is performed in interactive and semi-automatic modes. It is impossible to fully automate the picking process for real-life field seismic data, however, the program features a number of tools allowing to simplify and speed up this process as much as possible.

The picking mode is enabled by pressing the toolbar button. Use the left mouse button for picking the arrivals; right-click on a pick point removes the point.

To pick a sequence of points, use the *Multipicking* mode which is enabled by pressing the button. In the *Multipicking* mode, the traces between the specified start and end points are automatically picked using a selected picking algorithm. If the *Autopicking* mode (see below) is disabled, the picking is performed following a straight line between the specified start and end points.

The *Autopicking* mode, enabled by pressing the button, allows automatic picking using one of the specified search criteria. The search criterion (the signal phase you want to pick) is selected in the pop-up menu brought up by right-clicking on the *Autopicking* ( ) toolbar button. The available criteria are:

To null – search for the transition from null (or close to null) values in the specified window (the onset picking).

To extremum – search for the maximum absolute value in the specified window.

To maximum – search for the maximum value in the specified window.

To minimum – search for the minimum value in the specified window.

To best correlated point – search for the point of best correlation with neighboring traces.

Move to reciprocity point – if there is an assigned reciprocity point for a pick point, their times will be averaged.

If the *Multipicking* mode is enabled, the autopicking is performed for all traces in the specified trace range; otherwise, the search is performed for a single trace. It is important to choose the appropriate width of the time window (indicated with a circle or band) in which the search for the specified phase is performed. The width of the time window is adjusted using the mouse wheel with the Ctrl button pressed. In the *Multipicking* mode with *Autopicking* enabled, the preliminary picks points (in the form of grey circles) are displayed on the traces between the start point and the cursor position.

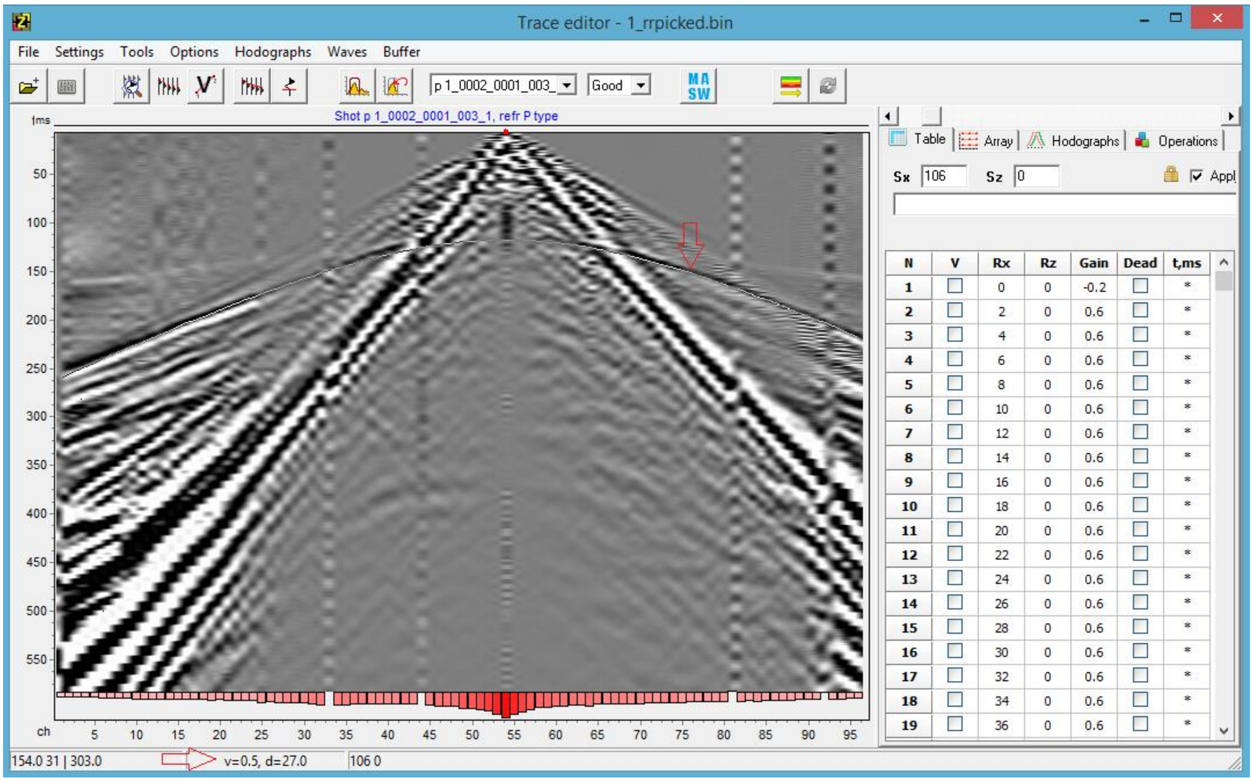
Note that the *Autopicking* mode is only suited for data of good quality.

The **Hodographs / Autocorrect current** menu command is used for automatic correction of the first arrival times based on the correlation of neighboring traces of the current pick.

One method of controlling the picking and data accuracy is comparing the reciprocal times which are indicated as individual circles on the current record. This routine is particularly effective when picking the data collected with dense observation systems, with a large number of source locations per spread. If there is a large discrepancy between reciprocal pick times, it is necessary to determine the reason, which could be either due to the picking of wrong arrivals (thus, comparing the reciprocal times controls the accuracy of picking) or due to erroneous shot (trigger) times. In the latter case, the shot times can be corrected automatically based on the reciprocal times, or manually. A typical picking process should be performed as follows:

1. Picking of individual shot gathers interactively or automatically, depending on the data quality.
2. Correcting individual picks based on the analysis of reciprocal times of the entire set of first arrival picks. The picking error can be controlled using the **Hodographs / Calc reciprocity error** command.
3. Correcting the shot times automatically for all shot gathers (**Hodographs / Correct Start time(reciprocity)**) or manually for each one individually.
4. Averaging of reciprocal times (**Hodographs / Average reciprocity data**).
5. Smoothing the picks, if necessary.

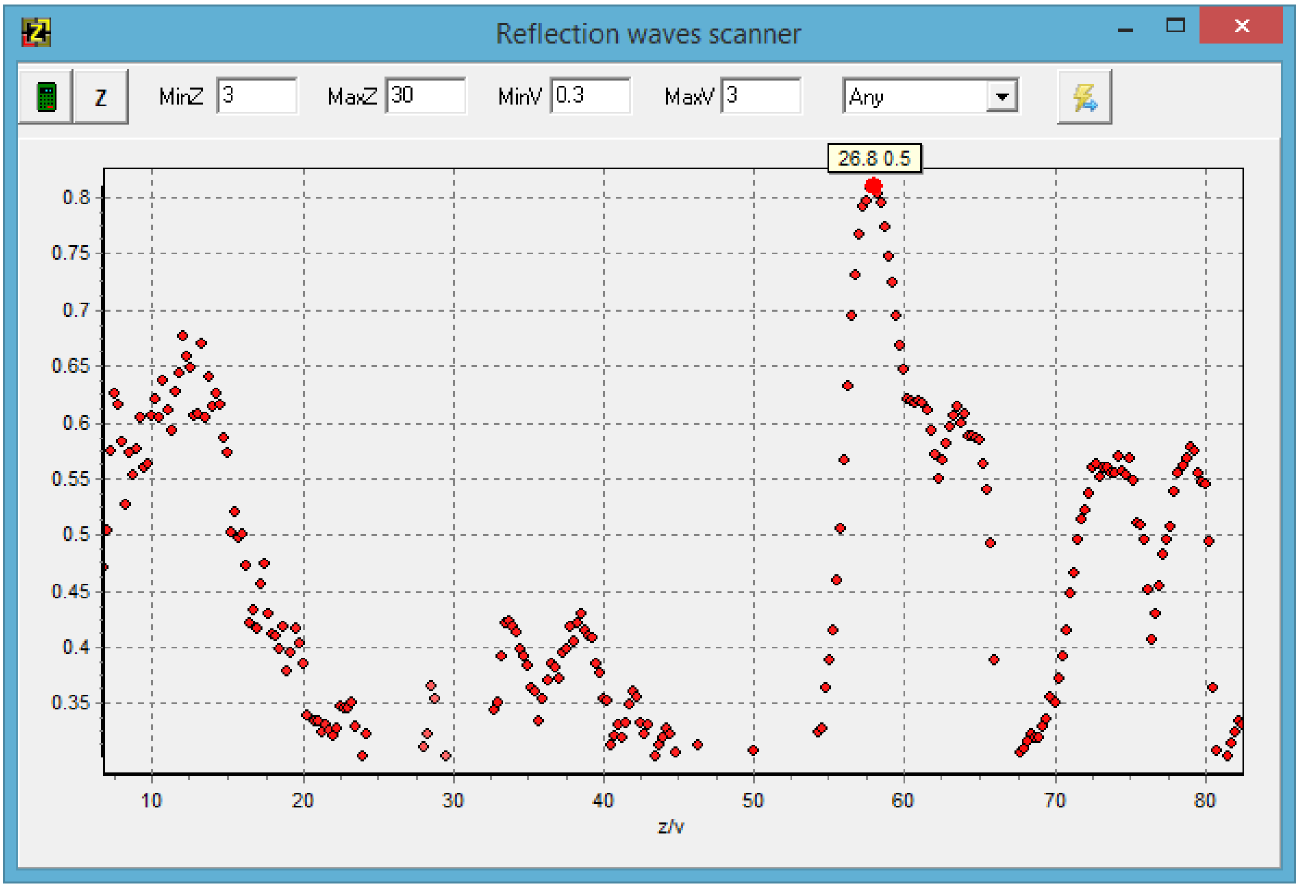
To pick arrivals of reflected waves, first select the wave type in the **Waves** menu section. Typically, shallow refraction data sets will only contain one reflection event, if any; therefore, it is recommended to select the **Reflected - 1** option. Reflected waves have travel time curves in hyperbolic shape.

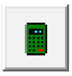


To pick reflection arrivals, press and hold down the V key after selecting the **Reflected** wave type; the hyperbolic travel time curve will be plotted on the seismic record in the interactive mode, following the mouse cursor position. The reflection parameters (velocity and depth) are displayed in the window status bar. The mouse wheel is used to change the shape of the hyperbola. After aligning the hyperbola with a reflection event, right-click to carry out the automatic picking of the entire gather. After automatic picking, the arrival times can be corrected manually.

When selecting a reflected wave type in the **Waves** section of the menu, the **Hodographs / Reflection scanner** command becomes available. This command opens a special module that performs the calculation and display of the probability of reflection events as a function of depth and velocity. On the toolbar of the module window, you can narrow the search range by setting the velocity and depth limits of reflection events (the **MinZ** and **MaxZ** input boxes for depth, **MinV** and **MaxV** – for velocity), or selecting the correlation criterion from the 

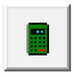
drop-down list (**Any** – any extremum, **Positive** – amplitude maximum, **Negative** – amplitude minimum).

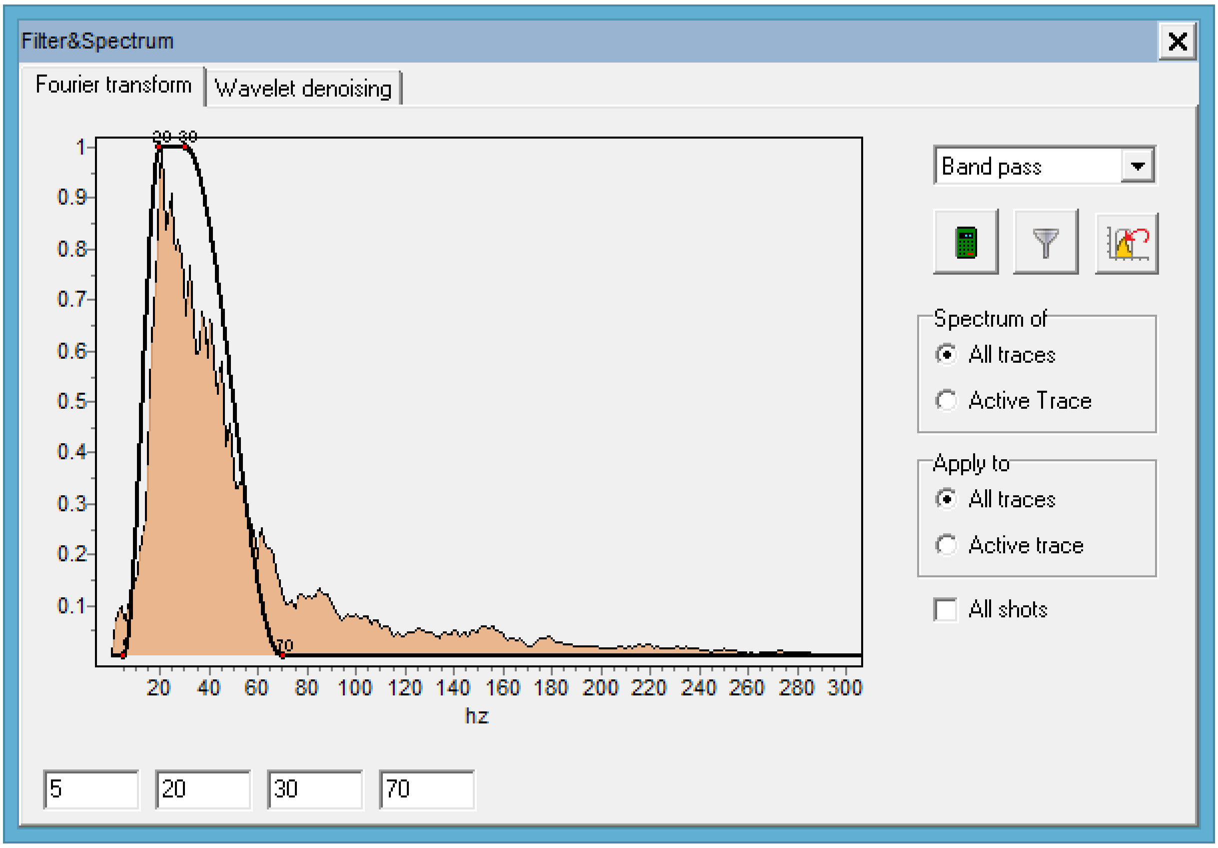


The  button calculates the probability in the selected range and plots the probability graph. The probability can be plotted as a function of depth (Z) or as a function of depth-to-velocity ratio (Z/V); the  button allows you to switch between the two plotting modes. When left-clicking on a point on the graph, the corresponding depth and velocity are displayed next to the point (Z V), and the corresponding travel time curve is plotted on the seismic record. To create a pick from the selected travel time curve, press the  button.

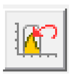
Data filtering

The data filtering module is opened using the **Tools / Filtering** menu command. The module is used for spectra analysis and filtering of seismic records. The module window consists of three tabs providing spectra calculation and filtering options for three types of filters: Fourier transform, wavelet transform and F-K filters.

The **Fourier transform** tab consists of a graphic area showing the frequency spectrum and the shape of the filter, and a set of options for the Fourier filter. To calculate the frequency spectrum for the specified working area of the current gather, press the  button. If the **All traces** option is selected, the average spectrum for the entire working area is calculated, otherwise, the spectrum is calculated for the active trace. You can change the scale of the spectrum plot by zooming in and out using the mouse (similarly to how it is done in the **Zoom** mode of the seismic record display), or by changing the scale of the axes (right-click on the axis with the Shift key pressed – **Options**).



The type of filter is selected from the drop-down list in the top left corner of the window. The available options are **Low pass**, **High Pass**, **Band pass** and **Band reject**. The filter is configured using the input boxes beneath the spectrum or using the mouse by dragging the nodes of the filter graph plotted over the spectrum.

After selecting the filter parameters, the filtering is performed by pressing the  button. The filtered seismic record is displayed immediately and the spectrum for the filtered record is recalculated. To return to unfiltered data, press the  button. If the **All shots** checkbox is checked, the filtering is applied to all seismic records in the project.

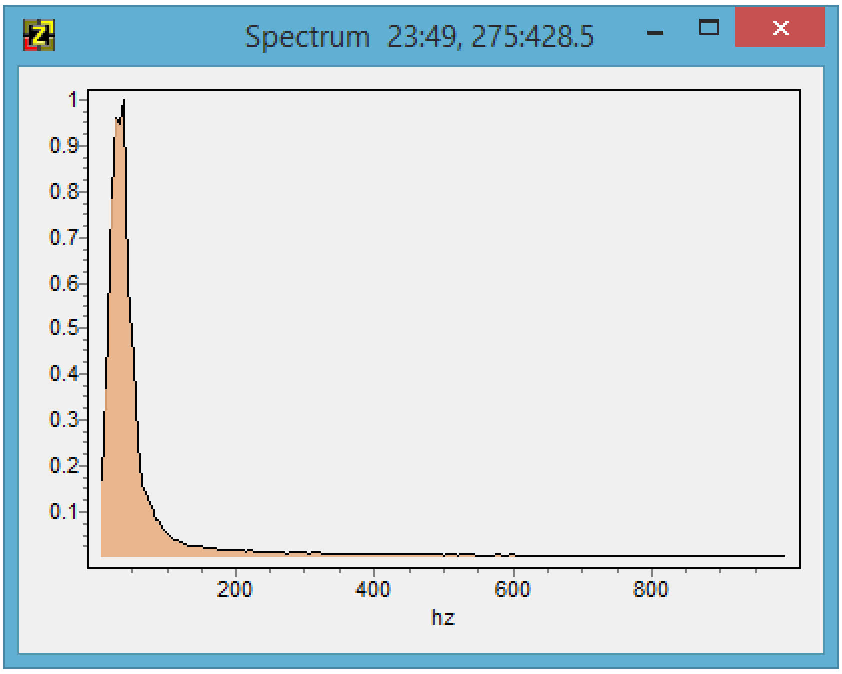
In the **Wavelet denoising** tab, you can perform wavelet transform filtering. The following filter configuration parameters are available:

Wavelet type – sets the wavelet type.

Wavelet order – defines the order of the wavelet (the complexity of its shape).

Decomposition level – determines the amount of fine details that will remain in the signal after filtering. The higher the value, the larger the features that are filtered out.

To see the frequency spectrum for a particular portion of the current seismic record, in the **Zoom** mode, select the area of interest with the mouse while holding down the Ctrl key.



In the window that appears, the spectrum for the selected portion of the record is displayed. The header of the window shows the selected trace and sample ranges.

MASW/ReMi module

The multi-channel analysis of surface waves (MASW) is a method based on dispersive nature of surface waves. Since surface waves contain a significant portion of generated seismic energy and their velocities are much lower than velocities of other types of waves, it is possible to reliably separate and process the surface wave portion of the wavefield.

Several modifications of the surface wave method were developed over the years, but generally, the surface wave data can be obtained using the active and passive techniques. For the active MASW, the fieldwork methodology is very similar to the seismic refraction method.

However, it should be kept in mind that the resolution and depth penetration of the MASW method are determined by the geometry of the array, record length, sampling frequency and geology.

The MASW method studies the dependence of phase velocity on frequency which can be visualized in the form of a dispersion curve. The lower the frequency for which the phase velocity is determined, the greater the penetration depth of surface wave energy. Obviously, the successful application of MASW also depends on the geology of the subsurface; the most favorable geological conditions are those with a smooth, laterally consistent increase in shear wave velocity with depth.

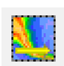
Extraction of dispersion curves from field data is the most demanding part of the processing routine, requiring certain skills and experience. Typically, the processing is performed in the frequency domain using the calculated KF or VF spectra. There are several basic algorithms for automatic and semiautomatic extraction of dispersion curves; **ZondST2D** uses the algorithm proposed by Dr. Choon Park, the author of the MASW method, in 1999.

In general, the velocity of surface waves is influenced by three parameters – compressional wave velocity, shear wave velocity and density, but only the shear wave velocity (Vs) has a tangible influence. Therefore, the interpretation of MASW data results in a set of shear wave velocity depth profiles.

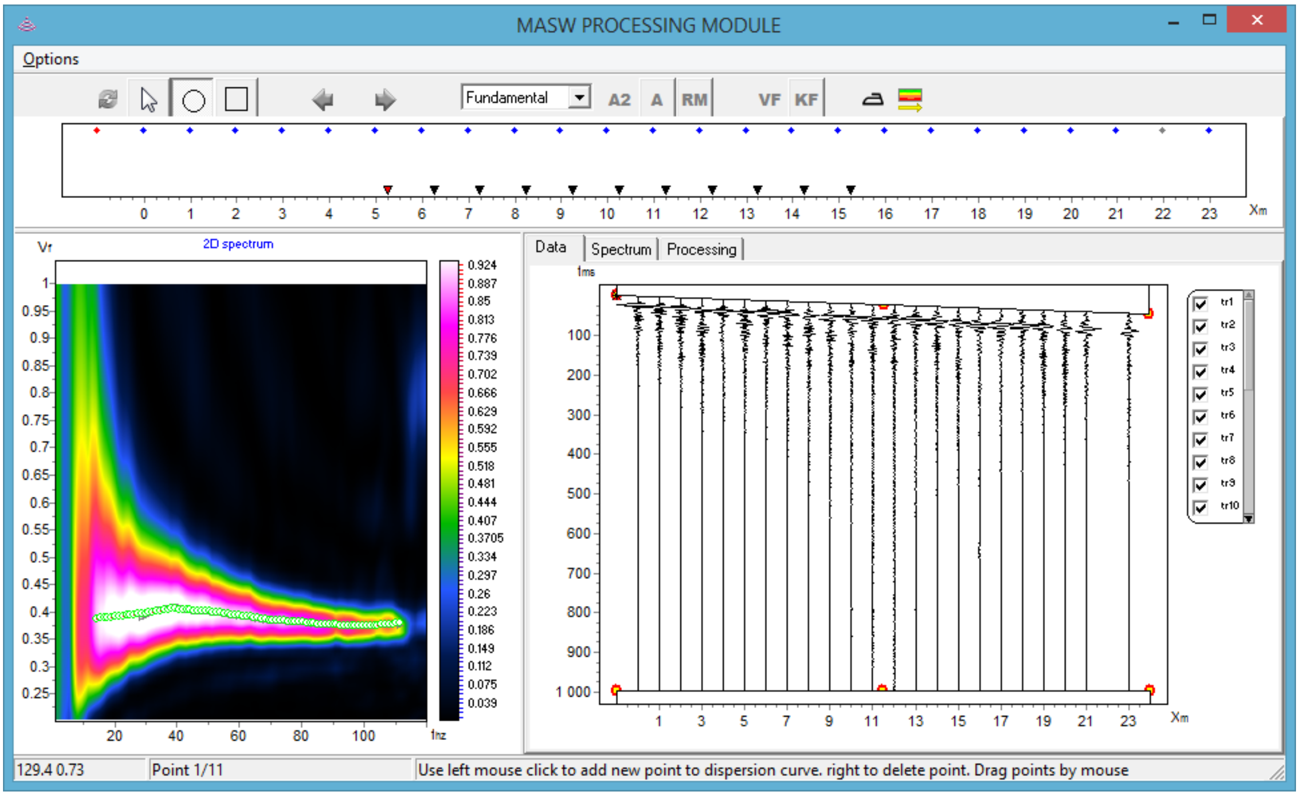
Summarizing, MASW surveys are carried out following the three main steps:

1. Obtaining field data. The survey design is based on the desired penetration depth and resolution. It is possible to use seismic refraction data sets for MASW processing.
2. Extraction of dispersion curves. This is the "interpretative" part. Depending on the subsurface conditions, data quality and other factors, different parameters of the two-dimensional spectrum calculation algorithm can be selected to improve the resolution.
3. Inversion of dispersion curves to obtain shear wave velocity depth profiles. A horizontally layered medium is used as the basic inversion model. In **ZondST2D**, the inversion algorithm is adapted to obtain smooth and piecewise smooth parameter distribution.

Processing of MASW data in **ZondST2D** begins with specifying the survey geometry. This procedure is performed in the **Trace editor** module. After the geometry is entered, the MASW module is started by pressing the 

button on the **Trace editor** module toolbar. Generally, the MASW module operates in two modes – *Picking* mode and *Inversion* mode; switching between the modes is done using the  and  toolbar buttons. The module starts in the *Picking* mode; after entering the module, the dispersion curve analysis window appears, divided into several sections.

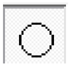
The MASW/ReMi window in the *Picking* mode consists of the **Options** menu, a toolbar, a graphic section showing the survey geometry, a graphic section showing the dispersion image and picked curve, and a multi-tab section.

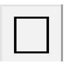


The module toolbar contains the following options:

 – recalculate two-dimensional spectrum with new parameters. The parameters are set in the **Processing** tab.

 – manual picking mode.

 – semi-automatic picking mode in which the search is performed in a circlular window. The size of the circle is adjusted with the mouse wheel.

 – semi-automatic picking mode in which the search is performed in a square window. The size of the square is adjusted with the mouse wheel.

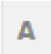
 – go to the previous MASW point. Depending on the selected array type (see the **Working with the module in Picking mode** section), some MASW points may appear empty,

i.e., shots collected with reverse array configuration (source coordinate is greater than receiver coordinates) when the *Forward* array type is selected.

 – go to the next MASW point.

– specify the mode being picked. **Fundamental** is the fundamental mode, **Mode 1** and **Mode 2** are the first and second higher modes. If higher modes are present in the spectrum, using them in the inversion increases the accuracy of the resulting model.

– display a two-dimensional spectrum of squared amplitude (it is sharper in appearance).



– display a two-dimensional amplitude spectrum.

– display the ReMi spectrum. The vertical gradient of the amplitude spectrum is displayed.

– display the velocity-frequency spectrum.

– display the wavenumber-frequency spectrum.

 – smooth the picks.

 – switch to the *Inversion* mode. You can switch between the inversion and picking modes at any time for adjusting the dispersion curves and models.

Description of the Options menu commands (Picking mode)

The **Options** menu of the module contains a number of additional options along with those duplicating the toolbar options.

Options	Delete active curve	Delete the dispersion curve for the current mode of the current dispersion image.
Options	Copy curves	Copy the current dispersion curve to the clipboard.
Options	Paste curves	Paste the dispersion curve from the clipboard to the current dispersion image.
Options	Clear muting	Reset the muting settings that limit the spectrum calculation area (the muting is configured in the **Data** tab).
Options	Spectrum	*Recalculate* – recalculate the spectrum with new parameters.

		KF spectrum – display the wavenumber-frequency spectrum. *VF spectrum* – display the velocity-frequency spectrum. *A style* – display a 2D amplitude spectrum. *A^2 style* – display a 2D spectrum of squared amplitude. *ReMi style* – display a ReMi spectrum. *Clip extremal values ReMi* – clip outliers in the data when calculating the ReMi spectrum.
Options	Picking	Enable the picking mode.
Options	Zoom&Scroll	Enable the **Zoom&Scroll** mode (allows zooming in and out on a portion of dispersion image).
Options	Smooth	Smooth the picks.
Options	Change mode	Allows switching between the picked modes by clicking on the corresponding dispersion curves.
Options	Extra settings	*Precise point mode* – enables precise picking mode. When using semi-automatic tools for picking (circle or square), the pick points are placed strictly at the maximum of the spectrum. *Apply to current mode* – if this option is enabled, actions (such as smoothing or deleting pick points) are performed only for the current mode (fundamental or a higher mode).
Options	Fundamental mode	Picking of the fundamental mode.
Options	Mode 1	Picking of the first higher mode.
Options	Mode 2	Picking of the second higher mode.
Options	Station positions	*Geometric center S-R* – the data point X coordinate is the average of all midpoints between the source and receiver coordinates (only traces participating in the spectra calculations are taken into account). *Geometric center R* – the data point X coordinate is the average of all receiver coordinates. *Autorecalculate* – automatically recalculate data point X coordinates when you change the set of participating traces or array type. *Edit position* – edit the data point X coordinates in a table

		opened in a new window.
Options	Units	Specify the axes units for seismic display: samples (*Index*) or time (*mSec*), channel number (*Channel*) or X coordinate (*Meters*).
Options	Export	*Curves to text* – export dispersion curves to a text file.
Options	Go to inversion mode	Switch to the *Inversion* mode.

Working with the module in Picking mode

The top section of the module window displays a graphic plot of the survey geometry for the current MASW point. By the MASW point, we will understand an array consisting of a source and a group of receivers (for the passive data – only a receiver array). The receivers can be on the right (forward array), left (reverse array) or on both sides of the source. Often it is required to adjust the size of the receiver group participating in the calculation of the two-dimensional spectrum. You should consider removing the traces with small offsets from the source which create a negative near-field effect due to distortion of the wave front, as well as the traces with large offsets which might create a far-field effect due to the increasing noise component in the data with distance. It is also recommended to remove dead traces.

In the bottom left of the module window, a two-dimensional spectrum (a dispersion image) is plotted. The dispersion image is a color plot of the signal amplitude as a function of phase velocity (wavenumber) and frequency. The picking of dispersion curves is carried out by the maximum values of the signal amplitude. Dispersion curves depend on the subsurface conditions in a complex way, have different shapes, and may contain several modes (several sections of the curve shifted relative to each other along the frequency axis). A fundamental mode, which represents the surface wave mode penetrating into the deeper layers, and higher modes, which represent the modes propagating closer to the surface, are distinguished. Most often, only the fundamental mode can be distinguished on the dispersion image. If higher modes are present, they can be picked and included in the inversion, which will improve the resolution and accuracy of the resulting model.

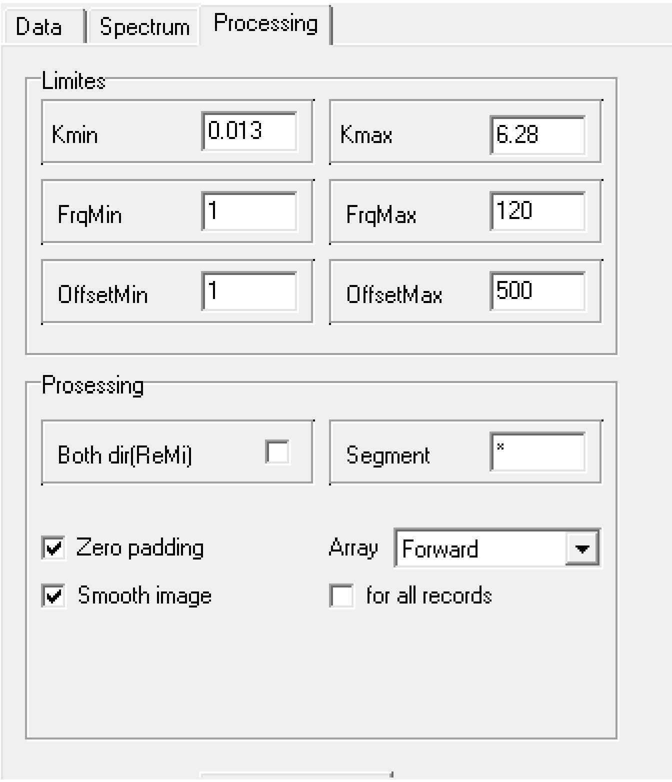
The dispersion curve picking is performed on the KF spectrum plot and can be carried out in the manual and semi-automatic modes. In the manual mode (the arrow tool on the toolbar), left-click on the dispersion image for picking; right-click on a pick point to remove it. In the semi-automatic mode (the circle and square tools on the toolbar), the search for maximum signal amplitudes is performed automatically in a circular or square window which size can be adjusted with the mouse wheel. In this mode, the pick sampling frequency depends on the sampling frequency of the field record. The mode of the active pick is selected using the 

drop-down list or the **Options** menu. Picks for different modes are displayed in different colors.

In the bottom right of the module window, there is a multi-tab section. The **Data** tab displays the current shot gather. You can disable the traces you want to exclude from the calculation of the 2D spectrum. To do this, double-click on the particular trace you want to exclude, or deselect it in the legend. The amplitude gain of an individual trace can be adjusted using the mouse wheel with the cursor hovering over the trace. A muting tool allows you to limit the area over which the spectrum is calculated, which can be used for suppressing body waves arriving at early times or noise at later times of the record. The two muting polygons at the top and bottom of the record can be reshaped by dragging the corresponding nodes with the mouse. The areas of the record covered with the polygons will not be used in the calculations.

The **Spectrum** tab displays the averaged one-dimensional frequency spectrum of the current seismic record. Two vertical sliders allow selecting the minimum and maximum frequencies to be used for the calculation of the two-dimensional spectrum.

The **Processing** tab contains the settings that control the 2D spectrum calculation algorithm.



The **KMin** (**VMin**), **KMax** (**VMax**), **FrqMin** and **FrqMax** input boxes set the limits of the calculated 2D spectrum. They are used for narrowing the search area and for enhanced spectrum representation. The minimum and maximum phase velocity values are specified based on the expected shear wave velocity range at the survey site.

The **OffsetMin** and **OffsetMax** values determine the minimum and maximum receiver offset in meters.

Both dir(ReMi) – used for the passive (ReMi) data in which case the source position is unknown.

Zero padding – allows obtaining a more detailed (frequency-wise) spectrum.

Smooth image – smooths the dispersion image to facilitate the picking.

In the **Segment** input box, you can specify the length (in samples) of time segments of the passive (ReMi) record to be processed. These segments will be processed individually; this speeds up the calculation process and allows obtaining more reliable spectra in some cases. If the * symbol is entered, the whole length of the record is used.

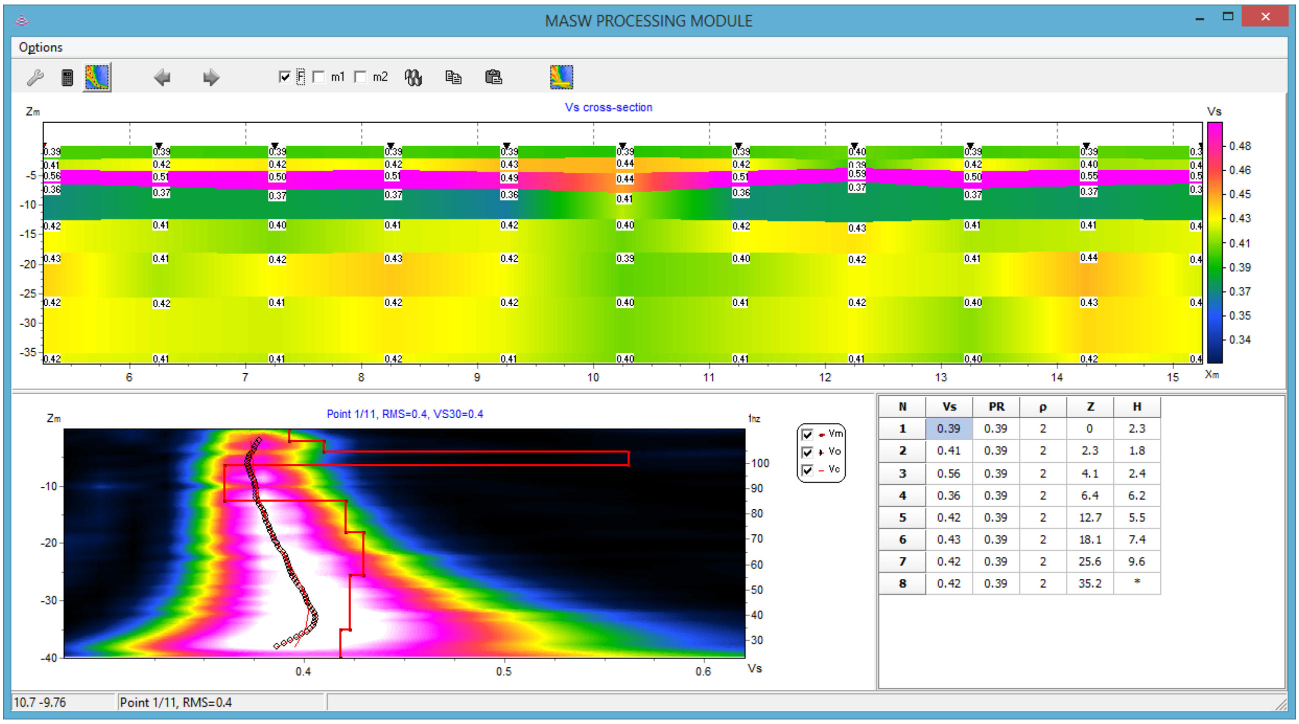
Array – defines the type of the MASW array. *Forward* – the source coordinate is less than the first receiver coordinate. *Reverse* – the source coordinate is greater than the last receiver coordinate. *Forward&reverse* – the receivers are located on either side of the source. When you change the type of array to *Forward* or *Reverse*, the program disables the traces with coordinates less or greater than the source coordinate, respectively (taking into account the values in the **OffsetMin** and **OffsetMax** boxes). The participating traces can also be disabled/enabled manually in the **Data** tab.

For all records – using this option, an average spectrum for all MASW points can be calculated. This allows you to get an idea of the background spectrum at the site but can take a considerable amount of time.

MASW Inversion mode

To go to the *Inversion* mode, press the  toolbar button or select the **Options / Go to inversion mode** command. You can switch between the picking and inversion modes at any time – not necessary to pick all the data at once; particular dispersion curves can be modified and inversion re-run.

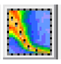
For each MASW point, a one-dimensional model (depth profile) is obtained through inversion of the dispersion curve by modeling the subsurface as a horizontally layered medium. A set of depth profiles along the survey line produces a shear wave velocity section (V_S section). In general, the goal of interpretation is to achieve a good match between the picked and computed dispersion curves for a geologically reasonable model. The parameters of the model layers are S-wave and P-wave velocities, density and thickness. Typically, the inversion model parameters are limited to the number of layers, their thickness and S-wave velocity.



The module toolbar in the inversion mode contains the following buttons:

 – open the dialog box for configuring the initial model and inversion parameters.

 – solve the forward problem for the current model.

 – show the dispersion image in the background.

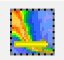
 – go to previous MASW point. If there is no dispersion curve for this point, the inversion cannot be performed.

 – go to next MASW point. If there is no dispersion curve for this point, the inversion cannot be performed.

 – run the inversion. Right-click on the button starts the inversion for the current and all subsequent points.

 – copy the current model to the clipboard.

 – replace the current model with the model from the clipboard.

 – switch to the *Picking* mode.

The **F**, **m1** and **m2** checkboxes indicate the mode being inverted (**F** for fundamental, **m1** and **m2** for higher modes). It is recommended to invert the fundamental mode first, then add higher modes to the inversion.

Description of the Options menu commands (Inversion mode)

The **Options** menu contains a number of additional options along with those duplicating the toolbar options.

Options	Settings	Open the dialog box for configuring the initial model and inversion parameters.
Options	Run Forward problem	Solve the forward problem for the current model.
Options	Run Inversion	Run the inversion.
Options	Global search	Run the annealing inversion algorithm.
Options	Auto Recalculation	Automatic recalculation of the forward problem solution when the model parameters change.
Options	Fast solver	Enable fast solving of the forward problem.
Options	Copy model	Copy the current model to the clipboard.
Options	Paste model	Replace the current model with the model from the clipboard.
Options	Poisson ratio	Set the Poisson's ratio as the second parameter of the model. It is preferred to use the Poisson's ratio because P-wave velocity (V_P) has little effect on the calculation results.
Options	Vp value	Set the P-wave velocity (V_P) as the second parameter of the model.
Options	Section	*Interpolate* – display the V_S section with a smooth parameter distribution. *Standard* – display the V_S section as a set of individual layered models for each MASW point.

Options	Background	*Load* – import a graphic (image) file as a background for the V_S section. *Show* – display the V_S section background. *Remove* – remove the V_S section background.
Options	Draw labels	Display the V_S values in the V_S section.
Options	Load borehole data	Import borehole data in the CRT file format.
Options	Remove boreholes	Remove borehole data.
Options	Export / Model to CAD	Export the interpretation results to an AutoCAD DXF file.
Options	Export / Model as XYZ	Export the V_S section to a text file containing three columns: X – distance along the survey line, Y – depth, Z – shear wave velocity.
Options	Export / Report to Excel	Export the interpretation results to an Excel spreadsheet in the form of report tables.
Options	Export / Pseudo boreholes	Export V_S depth profiles (models) in the form of columns. The pseudoboreholes can be used to present the results of refraction tomography and MASW surveys together. Pseudoboreholes can be opened in the refraction data interpretation module (the main module of the program).
Options	Export / MOD1D file	Export V_S depth profiles (models) to a MOD1D file that can be opened in other Zond programs. MOD1D files can also be imported into a Zond project in the form of borehole columns.
Options	Export / VS30 profile	Export V_S30 values for each MASW point to a text file.
Options	Import MOD1D/2D	Import a model from a MOD1D or MOD2D file into the V_S section. The model coordinates should be in the same coordinate system as the MASW survey line.
Options	Go to processing mode	Switch to the *Picking* mode.

Working with the module in Inversion mode

The workspace of the MASW module window in the inversion mode is divided into three sections. On the top, the V_S section is displayed. The model (V_S curve), picked and calculated dispersion curves, and dispersion image for the current MASW point are plotted in the bottom left section. The bottom right section contains the tabulated model for the current MASW point.

The inversion process can be carried out if there is a dispersion curve for a given MASW point, which is plotted in the bottom left section of the window. The MASW model parameters can be edited either manually (by editing the V_S curve with the mouse or in the table in the bottom right section of the window) or automatically through the inversion process. During the inversion, only V_S values and layer thicknesses are corrected. Therefore, the other model parameters should be specified prior to inversion. There are two ways of specifying the compressional wave velocity (V_P) – by direct editing of V_P values or by editing the Poisson's ratio values (PR). The second way is preferred since Poisson's ratio is stable and varies within a narrow range. To switch between the V_P and PR values, use the corresponding **Options** menu commands or double-click on the header of the third column of the table.

You can fix individual V_S and layer depth (Z) values (or the entire set of values) for inversion. To do this, double-click on the cell containing the value you want to fix; the cell will change its color to gray. To fix all V_S or depth (Z) values, double-click on the corresponding column header.

The top section of the window displays the shear wave velocity cross-section, which is obtained by linear interpolation of V_S depth profiles of individual MASW points. The color scale on the right represents the assignment of colors to the plotted data values.

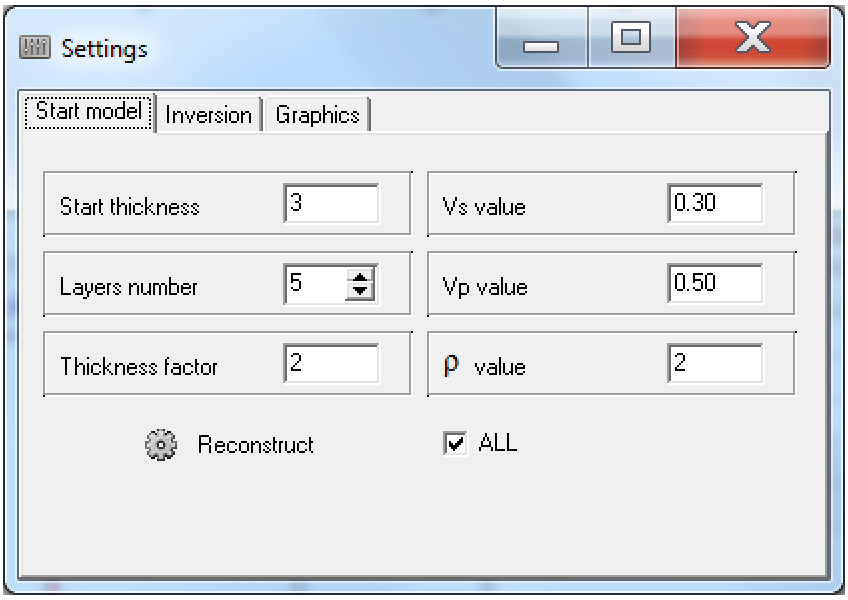
The MASW/ReMi picking data and interpretation results can be saved in the **Trace Editor** module as a BIN project file and in the main module of the program as an ST project file. The BIN format option makes it possible to save and open the data in the limited version of the program that supports only the surface wave mode.

*Note. It is possible to include MASW data in a joint inversion with refraction data in the layered model mode. To do this, activate the **Invert MASW** option of the **Layered** menu subsection of the main program module. You can evaluate MASW data misfits after the joint inversion by going back to the MASW module and looking at obtained models and calculation results.*

Interpretation settings

The  toolbar button and the **Settings** menu command open the dialog box for configuring the initial model, inversion parameters and graphic settings of the V_S section.

The **Start model** tab contains settings for configuring the initial model.



V_S, V_P, ρ – set the initial values of shear and compressional wave velocities and density.

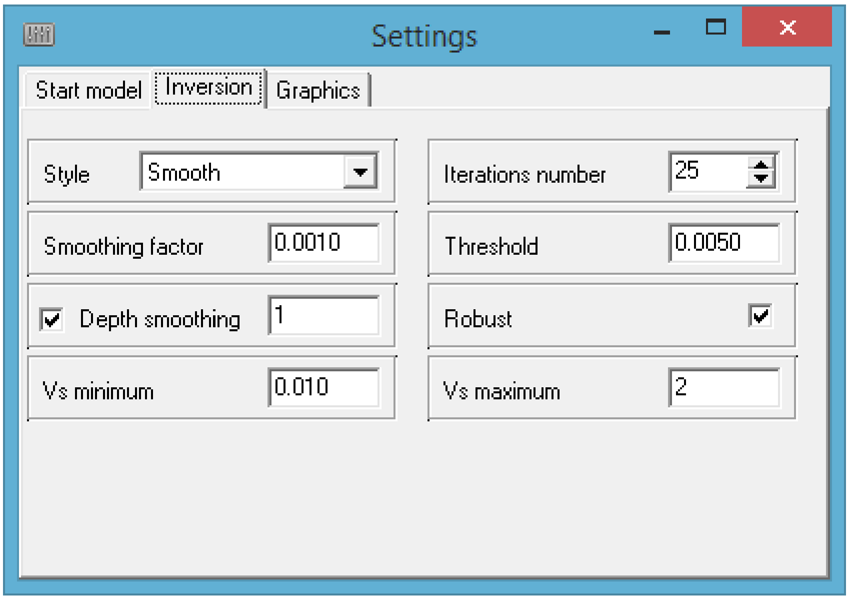
Start thickness – sets the thickness of the first layer.

Layers number – sets the number of layers.

Thickness factor – sets the thickness increment for each subsequent layer (1-5).

The **Reconstruct** button should be used after changing the parameters of the initial model; the previous model is replaced with the new one. If the **ALL** checkbox is checked, new initial models are generated for all MASW points, otherwise, only for the current point.

The **Inversion** tab contains settings controlling the inversion process.



Style – defines the inversion algorithm.

Standard is a classical least-squares inversion algorithm with a damping parameter regularization. When the number of model layers is small, this algorithm allows achieving minimal misfits.

Smooth is a least-squares inversion using a smoothing operator and additional contrast minimization. This algorithm results in the smoothest parameter distribution.

Focused is a least-squares inversion using a smoothing operator and additional contrast focusing. This algorithm results in a piecewise smooth parameter distribution.

Smoothing factor – sets the ratio between the minimization of data misfit and model misfit. For noisy data or to obtain smoother and more stable parameter distribution, relatively large values of the smoothing factor are chosen (0.05-0.1). Large values of the smoothing factor will typically result in larger misfits. The smoothing factor is used in the *Smooth* and *Focused* inversion algorithms.

Depth smoothing – sets the coefficient of smoothing with depth. It is used when the portion of the data characterizing the deeper portion of the model is of poor quality. The value is selected empirically.

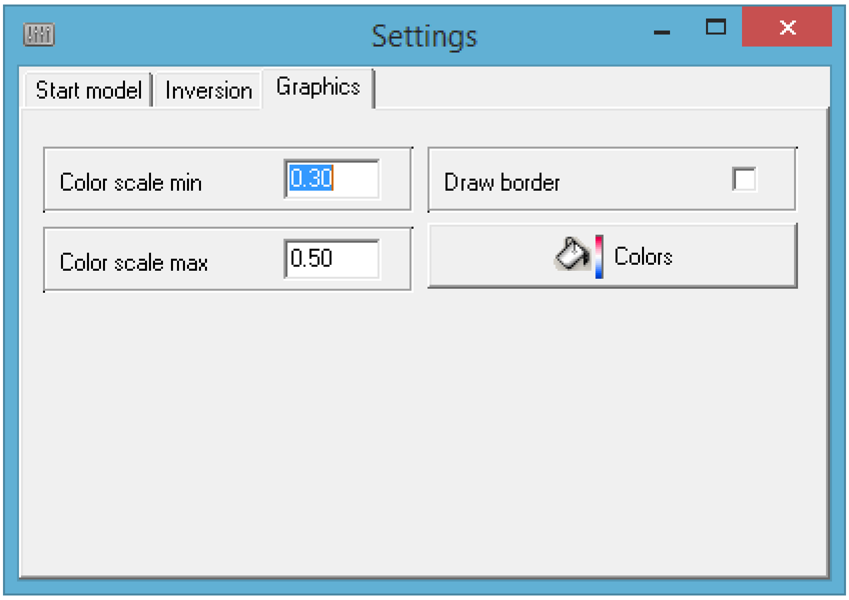
Threshold – sets the contrast threshold value of neighboring layers, after reaching which the parameters of the layers are not averaged. The value of this parameter is selected empirically in the 0.001-0.1 range. Choosing a very small value of this parameter can lead to algorithm divergence (in this case the value should be increased). Large values of the parameter lead to a smooth distribution. It is used in the *Focused* inversion algorithm.

Robust – this option should be enabled if there are spikes on the dispersion curves. If the number of bad points is comparable to the number of smooth points, this algorithm may not produce good results.

Iterations number – sets the number of inversion iterations.

The **Vs minimum** and **Vs maximum** input boxes set the shear wave velocity variation limits.

The **Graphics** tab contains graphic settings of the V_S section.



The **Color scale min** and **Color scale max** input boxes set the minimum and maximum V_S values of the color scale.

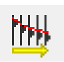
The **Draw borders** checkbox shows/hides boundaries between layers.

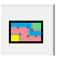
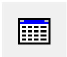
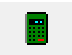
The **Colors** button opens the **Color Palette** dialog box.

Main window of the program

Main window toolbar

The toolbar is used for quick access to the most frequently used functions. It contains the following buttons:

	Open data or a project file.
	Open the **Trace editor** module.
	Open the dialog box for saving data or a project.

	Switch to mesh model mode.
	Switch to layered model mode.
	Switch to polygonal model mode.
	Open the **Program setup** dialog box.
	Open the **Data editor** table.
	Solve the forward problem.
	Start/stop the inversion.
	Run one-dimensional inversion. 1D inversion results can be used as an initial model for 2D inversion. Right-click on the button brings up a context menu where you can select the type of inversion: *1D solution* – 1D solution is calculated for a travel time curve averaged along the survey line, *1.5D solution* – the solution is calculated for individual travel time curves along the survey line.
	Undo changing the model.

Description of the main menu commands

To bring up a tip about a particular menu command, right-click on that command in the menu. The table below provides the description of menu commands.

File	Create synthetic survey	Open the module for creating a synthetic measurement system for modeling. You can specify the geometry of land surveys, borehole surveys, or a combination of both. After the module is closed, the program will run in modeling mode. This module is useful for planning seismic refraction surveys.
File	Open file	Open data or a project file.
File	Import data from text/excel	Import data from a multi-column text file or Excel spreadsheet. Column headers should be specified in the first row of the table.
File	Save file	Open the **Save As** dialog box.
File	Edit file	Open the data file opened in the program in the Notepad

		editor.
File	Print preview	Open the dialog box for printing the workspace.
File	Recent	Open one of the recently opened projects.
File	Russian	Switch the program language to Russian.
File	English	Switch the program language to English.
File	Exit	Exit the program.
Options	Project information	Show information about the currently opened project. This information can be edited.
Options	Mesh constructor	Open the dialog box for configuring the inversion model mesh. The constructor contains a set of options for automatic mesh creation, as well as advanced options for experienced users.
Options	Program setup	The main settings of the program. Most of the settings are for inversion.
Options / **Modules**	Data picker	Open the **Trace editor** module.
Options / Modules	Geological editor	Open the module for plotting an interpretive (geological) cross-section based on the interpretation of the refraction model. A polygonal object assignment interface is used.
Options / Modules	3D fence diagram	Open the module for three-dimensional visualization of seismic model sections based on their coordinates. Several model sections in MOD2D file format can be plotted in the module.
Options / **Inversion**	Set boundaries	Open the toolbar for specifying known boundaries (contacts) to be taken into account when performing the inversion. Use this tool if the exact position of the boundaries is known (e.g., from borehole data). Try to set the boundaries as close as possible to the model mesh nodes. It is recommended to use the following combination: *Occam inversion*, *Smoothing factor = 0.1-1*.

Options / Inversion	**Optimization**	A set of inversion optimization parameters.
Options / Inversion / Optimization	Optimization / Line search	Find the optimal damping factor at each iteration (if enabled, the inversion process will take approximately 3-4 times longer). In most cases, this option allows you to achieve the lowest RMS error in a reduced number of iterations, but it increases the time taken for each iteration and may result in the solution trapped in a local minimum. If the option is disabled, the damping factor is controlled by two parameters – **Smoothing factor** (initial value) and **Factor** (reducing factor).
Options / Inversion / Optimization	Optimization / Factor	If the **Line search** option is disabled, the **Factor** controls the degree of damping during the inversion. For the first iteration, the **Smoothing factor** value is used, and for each subsequent iteration, this value is divided by the user-specified **Factor**. The **Smoothing factor** can be determined automatically if the checkbox near the **Smoothing factor** input box in the **Model** tab of the **Program setup** dialog box is checked.
Options / Inversion / Optimization	Optimization / Lim based inv	If the overall parameter limits or individual limits for some cells are set too narrow, the inversion will try to exceed the parameter values beyond the set limits. This can greatly affect the rate of convergence. In this case, activating this option will reduce the contribution of cells that exceed the set limits, at the same time making such an exceeding difficult by applying special parameter normalization.
Options / Inversion	Resolution	A set of sensitivity factors used to increase the resolution of the inversion with depth. Increasing the sensitivity leads to increased influence of the lower cells of the model.
Options / Inversion	**Smoothness**	A set of options to control the smoothness and smoothing operator. These options can strongly influence the result of the inversion. They represent the second member of the

		target function C^C*(m-m0).
Options / Inversion	Smoothness / m0 - start model	m0 (reference model) is a user-defined model or inversion result. The main goal of inversion in this case is to reduce the RMS error while keeping similarity to the reference model. The degree of similarity is controlled by the **Smoothing factor** set in the **Model** tab of the **Program setup** dialog box. This option works with the *Occam* and *Focused* inversion methods.
Options / Inversion	Smoothness / m0 - median	**m0** (reference model) is the median of the model at the current iteration. The main goal of inversion in this case is to reduce the RMS error while keeping the model as smooth as possible. Smoothness is controlled by the **Smoothing factor**. This option works with the *Occam* and *Focused* inversion methods and is best suited for focused inversion and inversion with a priori boundaries.
Options / Inversion	Smoothness / m0 - previous model	**m0** (reference model) is the model obtained at the previous iteration. The main goal of inversion in this case is to minimize the RMS error while ensuring a stable convergence. The convergence rate is controlled by the **Smoothing factor**. This option works with the *Occam* and *Focused* inversion methods. The focused inversion sometimes might not yield the desired result (a piecewise-constant model).
Options / Inversion	Smoothness / Diagonal filter	Apply diagonal smoothing during the inversion. Use this option if there are inclined structures in the section.
Options / Inversion	Smoothness / Average window	Apply 2D moving average (median) filter to the reference model.
Options / Inversion	**Cross gradient** / Pushing factor	The main parameter of the joint inversion which controls the degree of similarity (minimum of cross-gradient) of the models of two different methods. A suitable value is chosen by trial and error in the range from 0 to 1000. If the value is set to zero, the input data sets are inverted independently.

		Larger values may result in higher RMS error for one or both of the methods. You can balance the misfits by assigning weights to the respective data sets.
Options / Inversion	Cross gradient / Off-layers num	Often the near-surface portion of the section is highly heterogeneous and variable in different ways for different geophysical methods. In such a case, several upper layers should be excluded from the cross-gradient operator. The near-surface portions of the models will then be inverted independently.
Options / Inversion	Cross-gradient / MinMax range	Set the parameter range for the second method involved in the joint inversion.
Options / Inversion	Cross-gradient / CC criteria	Use the common correlation criterion for joint inversion of two models. If this algorithm is activated, the **Pushing factor** should be set in the 0-2 range.
Options / Inversion	Cross-gradient / Seismic data vs	Select seismic data for joint inversion (e.g., S-wave data for joint inversion with P-wave data). When selected, the **Limited by PR** option appears, which controls that the Poisson's ratio stays within reasonable limits.
Options / Inversion	Cross-gradient / Gravity data	Select gravity data for joint inversion.
Options / Inversion	Cross-gradient / Magnetic data	Select magnetic data for joint inversion.
Options / Inversion	Cross-gradient / Anisotropy data	Determine the anisotropy during the joint inversion.
Options / Inversion	Cross-gradient / BG Image	Select an image as a base for image-guided inversion. It is recommended to use grayscale images.
Options / Inversion	**Underwater options**	A set of options for marine/aquatic surveys.
Options / Inversion	Underwater options / Velocity	Set the P-wave velocity in water.

Options / Inversion	Underwater options / Invert	Determine the velocity in water automatically during inversion.
Options / Inversion	Underwater options / Subnodes number	Set the number of subdivisions for the water column (5-10).
Options / Inversion	Underwater options / No direct waves	Use this options if no water arrivals (direct wave) are picked.
Options / Inversion	Invert start times MAX-10	Determine shot time (trigger) errors during inversion. This option should be enabled if it is difficult to make the shot time corrections manually.
Options / Inversion	Invert visible data	With this option enabled, only the displayed travel time curves will be inverted. In the legend box, you can disable the curves you want to exclude from the inversion.
Options / Inversion	Invert anisotropy	Determine the velocity anisotropy during inversion.
Options / Inversion	Long line inversion	This procedure splits the profile into several overlapping sections which are inverted independently. For most cases (if the number of receivers does not exceed 10,000), this procedure is not recommended.
Options / **Data**	First break times	Display first arrival times.
Options / Data	Apparent velocity	Display apparent velocities.
Options / Data	Travel time curves	Show travel time curves.
Options / Data	Iso S-R spacing	Show iso-offset graphs (points with the same distance between source and receiver).
Options / Data	Xpos=(S+R)/2	If this option is enabled, the travel time curve X coordinate is calculated as a midpoint between the source X and receiver

		X; if disabled, the receiver coordinates are used.
Options / Data	Ray paths	Show the ray paths connecting the sources and receivers, calculated for the current model. The forward problem should be solved beforehand.
Options / Data	Data editor	Open the **Data editor** table.
Options / Data	Survey pseudosection	Open a pseudo-section of apparent velocities in a separate window.
Options / Data	Velocity offset plot	Open a plot of apparent velocities as a function of offset in a separate window. This function can be used when creating the initial model.
Options / Data	Display with shifts	Show the travel time curves with corrected shot times (time-shifted).
Options / Data	Display error gates	Show confidence intervals of picks. They are set in the **Trace editor** module using the pick quality option on the toolbar.
Options / Data	Smooth data	Smooth the first arrival or amplitude data, depending on the selected mode.
Options / **Model**	Block-section	Display the mesh model in the form of blocks (cells).
Options / Model	Smooth-section	Display the mesh model in smooth interpolated palette mode.
Options / Model	Contour-section	Display the mesh model in the form of contours.
Options / Model	Velocity	Display velocity in the model section.
Options / Model	Sensitivity	Display sensitivity function in the model section. This option is available in the *Contour-section* mode.
Options / Model	dVelocity	Display the full velocity gradient in the model section. This option is available in the *Contour-section* mode.

Options / Model	Quality	Display the quality function in the model section. The quality is determined as the number of rays passing through a cell normalized to the inversion misfit. This option is available in the *Contour-section* mode.
Options / Model	Display	*Velocity* – switch to the velocity mode. *Anisotropy* – switch to the velocity anisotropy mode. *Attenuation* – switch to the wave attenuation mode. In this case, the amplitude curves are displayed in the top section of the window instead of the travel time curves.
Options / Model	X:Z=1:1 scale	Set proportional (1:1) scaling of the model section.
Options / Model	Model editor toolbar	Open the toolbar for editing the mesh and the model.
Options / Model	Extend bottom	If the section contains topography, this option extends the last model layer to the bottom edge of the model section.
Options / Model	Cut by rays	Trim the portions of the model not covered by the calculated ray paths.
Options / Model	Bound by cable	Trim the portions of the model that extend beyond the receiver array.
Options / **Topography**	Topo coefficient	Set the coefficient for topography distortion with depth. If the survey line contains topography, the coefficient specifies the rate with which the mesh flattens out with depth. If it is set to 0, each new layer of the mesh has the same geometry as the first one, i.e. the geometry of the last layer matches the topography. If set to 1, the last mesh layer is horizontal.
Options / Topography	Import topography	Read in topography data from a text file. The file should contain two columns (X and Y).
Options / Topography	Remove topography	Delete the line's topography data. This option is used for testing purposes.
Options /	Restore	Restore the previously removed topography data.

Topography	topography	
Options / Topography	Edit topography	Edit topography in a table mode. The topography data can be copied from an Excel spreadsheet.
Options / Topography	Smooth topo	Smooth the topography profile by averaging the neighboring values.
Options / Topography	Suppress topo	Reduce the data to flat surface topography through inversion. The topography data is removed from the project after the procedure.
Options / Topography	Set by mouse	Draw the topography profile on the model section using the mouse. This mode is similar to adding a priori boundaries.
Options / Topography	Splined intermediate	If this option is enabled, spline interpolation is used to calculate the elevations of nodes between the topographic points, otherwise, linear interpolation is used. The option is available when the project contains topography.
Options / Topography	Reverse line	Flip the survey line horizontally from left to right.
Options / Topography	Shift line	Shift the survey line horizontally by a specified value (in meters).
Options / **Boreholes**	Create/Edit borehole data	Add and edit borehole data (columns and logs).
Options / Boreholes	Load borehole data	Open a file containing borehole columns and/or logging data, or a MOD1D file (one-dimensional interpretation results).
Options / Boreholes	Remove borehole data	Remove the borehole data from the project.
Options / Boreholes	Set column width	Set the width of the borehole columns in the model section (in percentage of the survey line length).
Options / **Extra**	Model smooth/raster	This tool allows you to smooth the model or a portion of it, or group cells into blocks. The tool can be used for smoothing the strongly heterogeneous top of the model or for smoothing the *Blocks* inversion results.

Options / Extra	Display t0 map	Build an isoline map of delay times (t0) at each node of the current model.
Options / Extra	Data&model histogram	Open a chart showing the distribution of apparent (observed) and true (modeled) velocities. The window workspace allows you to set the minimum and maximum of the color scale.
Options / Extra	Build start model	Build an initial model based on apparent velocities for cross-hole tomography.
Options / Extra	Add extra nodes	Add additional nodes along the edges of the model. This option might be useful for cross-hole surveys. The option should be activated before opening a data file.
Options / Extra	Display receiver RMS	Display RMS error for each receiver/source.
Options / Extra	Orientation	Choose the arrangement of the workspace sections: vertical or horizontal. The vertical mode is convenient for working with the borehole seismic data.
Options / **Import/Export**	**Other modules**	A set of options for fast exchange of models between program modules.
Options / Import/Export	Other modules / From layered as model	Embed an arbitrary layered model into a mesh model.
Options / Import/Export	Other modules /From MASW as boreholes	Show the results of MASW interpretation in the model section in the form of boreholes.
Options / Import/Export	Other modules /From MASW as model	Embed an MASW V_S section into a mesh model.
Options / Import/Export	Import model/data	Open a data file with a graph (the file should contain two columns, X and Y) or a MOD2D file to display the graph/model in a separate window.
Options /	Remove	Remove the imported graph/model from the project.

Import/Export	model/data	
Options / Import/Export	**Model parts** / Save selection	Save a selected portion of the model to a text file (in the *Block-section* mode).
Options / Import/Export	Model parts / Load selection	Load a portion of the model from a text file and insert it into the model (in the *Block-section* mode).
Options / Import/Export	Model parts / Extract 1D log	Export a 1D model for a specified position to a text file.
Options / Import/Export	Model parts / Load 1D log	Load a 1D model (a velocity depth profile) from a text file and insert it into the model.
Options / Import/Export	**Background image** / Load image	Load a background image in the following formats: BMP, PNG, SGY, SEC. SEC is an internal **Zond** format which contains a spatial reference for an image (coordinates of the image corners).
Options / Import/Export	Background image / Remove background	Remove the background image from the model section.
Options / Import/Export	Background image / Change sizes	Change the size and position of the background image.
Options / Import/Export	Background image / Create shaded map	Use the shaded relief map based on the current model as a background.
Options / Import/Export	Load 1D model	Import a 1D model from a text file and embed it into the current model.
Options / Import/Export	Load MOD1D/2D	MOD1D and MOD2D are internal Zond formats. This option allows you to open models created in other Zond programs and projects. The imported model is embedded into the current model.
Options / Import/Export	Save MOD1D/2D	Save the model in the MOD1D or MOD2D format for future use in other Zond programs and projects.

Options / Import/Export	Import SeisOpt picking	Import first arrival data from a SeisOpt program file.
Options / Import/Export	Export model to Excel	Export the current model to an Excel spreadsheet.
Options / Import/Export	Export model to CAD	Export the current model (contour section in vector representation) to a DXF CAD file.
Options / Import/Export	Export model to SEG-Y	Export the current model to the SEG-Y format.
Options / Import/Export	Export model to Geosoft	Export the current model to the Geosoft Inc. generic format.
Options / Import/Export	Export ray paths	Export the ray path diagram to a text file.
Options / Import/Export	Draw model in Surfer	Plot the current model in the Surfer program. This option is available in the *Contour-section* mode. It might not function properly if several versions of Surfer are installed or exchange libraries are not installed.
Options / **Graphics**	Observed graphics	Open the dialog box for configuring the observed data graphs.
Options / Graphics	Calculated graphics	Open the dialog box for configuring the calculated data graphs.
Options / Graphics	Smooth contours	Smooth the model contours in the *Contour-section* display mode.
Options / Graphics	Smoothness	Set the degree of smoothness of the model contours. The greater the smoothing parameter, the smoother the contours.
Options / Graphics	Isolines high quality	Render contours in high quality. It is recommended to use this option only prior to printing/exporting the results because it slows down the program performance.
Options / Graphics	Bitmap output settings	Open the dialog box for setting up export image parameters.
Options /		If the project contains amplitude picks, this menu subsection

Attenuation		appears when switching to the attenuation mode (**Model / Display / Attenuation**).
Options / Attenuation	Modeling mode	Enable the amplitude modeling mode for the current ray coverage. For modeling, it is assumed that the amplitudes of all sources are equal to one.
Options / Attenuation	Frequency	Set the frequency of the wave which amplitudes are collected in the attenuation mode.
Options / Attenuation	Max offset	Set the offset limit above which the traces do not participate in the inversion of amplitudes (used in the case of noisy data).
Options / Attenuation	Joint-velocity	Use cross-gradient minimization with the current velocity model during the inversion of amplitudes.
Options / Attenuation	Invert sources A	This option can be disabled if the source produces the same amount of energy at different shot locations (e.g., for marine surveys).
Options / **GraviMagnetic**	Load new data	Read in gravimagnetic data from a multi-column text file. The first row should contain column headers. The data must be in the same coordinate system as the current model.
Options / GraviMagnetic	Add new data	Add gravimagnetic data to the project.
Options / GraviMagnetic	Field setting	Set up gravity and magnetic field parameters.
Options / GraviMagnetic	Subtract median grav	Subtract the median value from the measured gravity data to obtain the anomalous field.
Options / GraviMagnetic	Subtract median mag	Subtract the median value from the measured magnetic data to obtain the anomalous field.
Options / GraviMagnetic	Inversion	Invert gravity and magnetic data in polygonal mode.
Options / GraviMagnetic	Invert gravity	Enable inversion of gravity data in layered model mode.

Options / GraviMagnetic	Invert magnetic	Enable inversion of magnetic data in layered model mode.
Options / GraviMagnetic	Set weight of gravity	Set the weight of gravity data. Assigning weights allows you to control joint inversion misfits.
Options / GraviMagnetic	Set weight of magnetic	Set the weight of magnetic data. Assigning weights allows you to control joint inversion misfits.
Options / GraviMagnetic	Display GM window	Open a window with gravimagnetic data and the model (in the cross-gradient mode).
Options / **TDEM data**	Load TDEM data	Import TEM data in TDF (**ZondTEM1D**) or USF (*universal sounding format*) text formats. The coordinates should be in the same coordinate system as the current model. Joint inversion is available only for layered model mode.
Options / TDEM data	Show TDEM data	Open a window displaying the TDEM/FDEM data.
Options / TDEM data	Invert TDEM data	Include the TDEM/FDEM data in joint inversion in layered model mode.
Options / TDEM data	Set weight of TDEM	Set the weight of TDEM/FDEM data. Assigning weights allows you to control joint inversion misfits.
Options / **VES data**	Load VES data	Import vertical electrical sounding (VES) data in Zond text format. The coordinates should be in the same coordinate system as the current model. Joint inversion is available only for layered model mode.
Options / VES data	Show VES plot	Open a window displaying the VES data.
Options / VES data	Invert VES data	Include the VES data in joint inversion in layered model mode.
Options / VES data	Set weight of VES	Set the weight of VES data. Assigning weights allows you to control joint inversion misfits.
Options / **MT data**	Load MT data	Import MT data in Zond text format. Station coordinates (distances in km) should be in the same coordinate system as

		the current model. Joint inversion is available only for layered model mode.
Options / MT data	Show MT plot	Open a window displaying the MT data.
Options / MT data	Invert MT data	Include the MT data in joint inversion in layered model mode.
Options / MT data	Set weight of MT	Set the weight of MT data. Assigning weights allows you to control joint inversion misfits.
Buffer		The buffer allows you to store up to five models obtained using different inversion parameters. The models can be opened in a separate window for comparison.
Buffer	Model 1-5	Save/load the model into/from the buffer.
Buffer	Open	Open a window displaying all buffer models. This option is useful for comparing inversion results obtained with different settings.
Waves	P-refracted	Switch to the refracted P-wave mode.
Waves	S-refracted	Switch to the refracted S-wave mode.
Waves	Reflected-1	Switch to the reflected wave mode (first boundary).
Waves	Reflected-2	Switch to the reflected wave mode (second boundary).
Waves	Reflected-3	Switch to the reflected wave mode (third boundary).
Waves	Summary plot	Open a window displaying the summary plot of velocity, anisotropy and attenuation distribution models (if the latter were calculated using the corresponding modules). The window is divided into three sections where the selected parameter models can be plotted.
Help	About	About the program
Help	Manual	Open the program manual.
Help	Check for updates	Check for updates.

Help	ERROR!!! Set default values	Reset all settings and return to the default settings after restarting the program.
Help	Bing maps api_key	If a Bing Maps image is not downloaded automatically from the Internet, you should enter an actual Bing API key.
Help	Show news	Show news.
Help	Send message to us	Send a message to the developer. Only messages written in the Latin alphabet can be sent (use transliteration if sending messages in Russian).

When you switch to the layered model mode (the button on the main window toolbar), the **Layered model** sub-menu becomes available with the following options:

Layered model / Model constructor	Open the layered model constructor.
Layered model / Save to mesh	Embed the layered model into a mesh model.
Layered model / Load from mesh	Use the average value of all cells comprising a layer as the layer's parameter value.
Layered model / Invert VP&VS	Joint inversion of V_P and V_S data using a common geometry of layers.
Layered model / Invert boundaries	Determine the geometry of layers during the inversion (sometimes you only need to determine parameter values, e.g., when the boundaries are known and fixed).
Layered model / Invert MASW	Include MASW data in joint inversion in layered model mode.
Layered model / Invert MASW-VP	If this option is enabled, the current P-wave velocity model will be used in joint inversion with MASW data.
Layered model / Draw labels	Display parameter values in the nodes. The choice of the parameter to be displayed is made in the model constructor.
Layered model / Transparent	Do not fill layers. This allows you to see the results of the inversion in the mesh mode in the background and set the optimal

	initial model.
Layered model / Edit mode	Enable editing mode of the layered model. Editing is done using the mouse. The boundary nodes can be dragged in the vertical direction; right-clicking on the layer's label allows editing the parameter value.
Layered model / Save layers	Save the layered model to a text file.
Layered model / Load layers	Load a layered model from a text file.
Layered model / Create from boreholes	Create model boundaries using the boreholes loaded into the project.

When you switch to the polygonal modeling mode (the button on the main window toolbar), the **Modeling** sub-menu becomes available with the following options:

Modeling / Get values from mesh	Assign parameter values to polygons automatically. The value will be equal to the average value of all model cells within the polygon.
Modeling / Set values to mesh	Embed the polygonal model into a mesh model.
Modeling / Save polygons	Save polygons to a text file.
Modeling / Load polygons	Load polygons from a text file.
Modeling / Remove all polygons	Remove all polygons.
Modeling / Display color scale	Show the color scale bar next to the model section.
Modeling / Colors from color scale	Assign colors to polygons according to the color scale.

Status bar

The status bar is located at the bottom of the main program window and is divided into several sections containing various information:

• Coordinates of the mouse pointer and the active cell.

• Parameter value of the active cell.

• Model editor operating mode.

• Process indicator.

• RMS error.

• Additional information, e.g., a number of data points and model cells, or the inversion status.

Hotkeys

Arrow keys or mouse pointer in the model editing mode	Select active model cell.
Delete / cursor in the model editor	Reset the cell parameter to its initial value.
Insert / cursor in the model editor	Assign the current value to the active cell.
F / cursor in the model editor	Fix the value of the active cell.
X / cursor in the model editor	Select cells with similar parameter values (the “Magic Wand” tool).
V / cursor in the model editor	Remove selection.
Up/Down arrow keys after left-clicking on the color bar	Change the current value.
Space	Solve the forward problem.

Creating a synthetic mesurement system

The **File / Create synthetic survey** menu command opens a module for creating a synthetic measurement system for modeling.

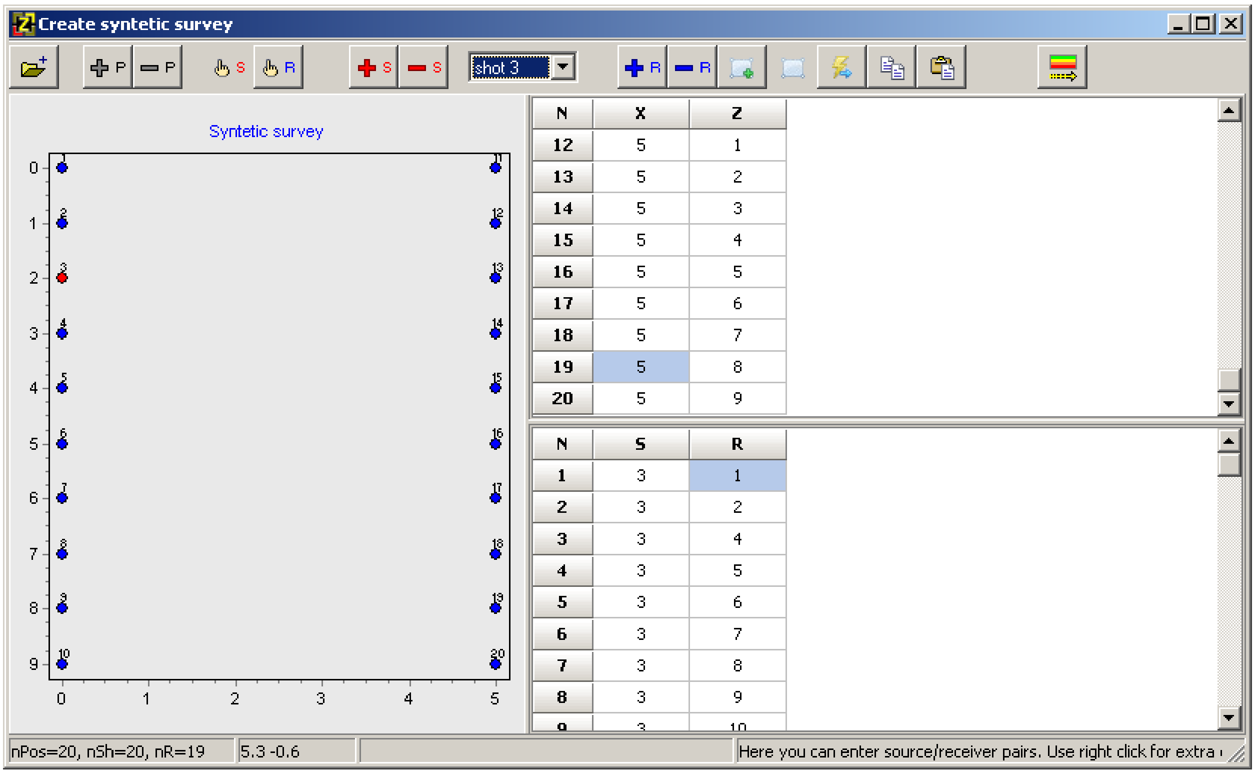
The module window is divided into three sections: a graphical representation of the created measurement system (left), a table containing horizontal and vertical coordinates of sources/receivers – coordinate table (top right), and a table displaying source and receiver IDs for a selected shot – shot table (bottom right).

The measurement system creation process generally consists of two steps – adding a required number of points with unique coordinates that can be used as source locations, receiver locations, or both, and then grouping them into shots. In the modeling that follows the creation of the measurement system, each shot corresponds to a travel time curve.

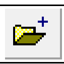
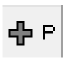
The graphic section on the left shows schematically the unique positions of sources/receivers in the form of small circles, and their IDs (column **N** of the coordinate table). Specifying a source/receiver group for a shot can be done using the mouse. The source of a current (selected) shot is displayed in red, the receivers – in blue.

The coordinate table contains the horizontal (**X**) and vertical (**Z**) coordinates of sources/receivers, which can be edited. By right-clicking on a table cell, you can specify a coordinate increment for all subsequent cells.

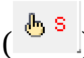
The shot table contains IDs of a source (**S**) and receivers (**R**) for the current (selected) shot, which can be edited.



The toolbar contains the following buttons:

	Open a file with survey geometry.
	Save the current survey geometry to a file.
	Add a new point (source/receiver position). The new point appears in the graphic section and in the coordinate table. The current number of unique positions is indicated in the status bar at the bottom of the module window.
	Delete the source/receiver position selected in the coordinate table.

	Enable source selection mode. Select a source point for the current shot in the graphic section using the mouse. After selecting a source, the program automatically switches to receiver selection mode.
	Enable receiver selection mode. Select a receiver point for the current shot in the graphic section using the mouse. If a point is selected while holding down the Shift key, the point is added to the receiver group (a new line appears in the shot table); otherwise, a new position is assigned to the current receiver (the selected line in the shot table).
	Add a new shot. After you press the button, the program switches to the source selection mode. A new item appears in the drop-down list on the right – **shot ***; the * symbol is replaced with the point ID after source position is specified.
	Remove the current shot.
shot 1	A drop-down list for selecting a current shot.
	Add a new source/receiver pair to the current shot (a new line in the shot table).
	Remove the current source/receiver pair from the shot (the selected line in the shot table).
	Select a group of receivers to be added to the current shot using the rectangle selection tool in graphic section.
	Select a new group of receivers for the current shot using the rectangle selection tool in graphic section.
	Brings up a pop-up menu with the following commands: *Select all positions* – select all points as receivers for the current shot, except the current source point. *Deselect all positions* – remove all receivers from the current shot. *Create full array* – create a measurement system with a source at each point and a group of receivers in a shot consisting of the rest of the points.
	Copy the IDs of the current shot receivers to the clipboard.
	Paste the receiver IDs from the clipboard to the current shot.
	Create a measurement system and proceed to modeling.

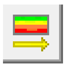
The suggested procedure of creating a measurement system is as follows. After opening the module, add the required number of unique source/receiver positions (points) using the button and specify their coordinates in the coordinate table (right-click on a cell to specify a coordinate increment for all subsequent cells). Then add the required number of shots (travel time curves) using the  button. For each shot specify a source position by clicking on a specific point in the graphic section in the source selection mode (), or entering the point ID in the S column of the shot table. For each shot specify a group of receivers by selecting points in the graphic section in the receiver selection mode (), or entering point IDs in the R column of the shot table. The easiest way to select a receiver group is by using the rectangle selection tools or commands from the  pop-up menu.

After configuring the measurement system, press the  button to proceed to modeling.

Synthetic measurement systems and created theoretical models can be used for testing purposes, e.g., for assessing the resolution of a measurement system. To use the data calculated for a created theoretical model and a given measurement system as observed data, the data should be saved into a file as **Zond calculated data [*.st]** type. The saved file can then be opened as an observed data file. Alternatively, the **Zond model with calculated [*.st]** file type can be used, but after opening the file the imported model has to be cleared before inversion (a new initial model created). It is then possible to perform inversion of the data and analyze the differences between the inverted model and the original one. This mechanism allows to conduct various experiments and test different measurement systems.

Working with data

Opening a data file

To start 2D interpretation of refraction data in **ZondST2D**, you need the first arrival data loaded into the project either by picking the first arrivals in the **Trace editor** module and switching into inversion mode by pressing the  toolbar button or by opening an ST data file containing survey geometry, topography and arrival times.

The text data files formatted in the **ZondST2D** program format have the ST extension (see the ST section for details**)**. Typically, one file contains data for one survey line. Data can also be imported from an arbitrary text file or Excel spreadsheet using the **File / Import from txt/excel** menu command. When selecting the **File** / **Open file** menu command you will be prompted to select one of the following file formats:

Zond data file [*.st]	Open data file or project file in Zond format.
Program configuration [*.cfg]	Open file with program configuration settings.
Reflex picking [*.tom]	Open data file in ReflexW format.

Note. To be opened correctly, the data file should not contain the following:

- non-conventional delimiters separating entries in a string (use Tab or Space characters);
- absurd values of measured parameters.

Preferably, the total number of measurements in one data file should not exceed 100000, and the number of unique source/receiver positions should not exceed 5000. The geometric units are meters, time – milliseconds.

ST file format

ST is an internal Zond file format. ST files can have a different structure – they are either data files that can be created and edited in a text editor, or project files created by the program when saving a project. The data files are text files containing information about the survey geometry, topography, arrival times and amplitudes (the structure of such a file is described below). Project files are binary and can only be read with the **ZondST2D** program. They contain all the project information, i.e. observed data, calculated data, models, stored a priori information, data from other methods, etc.

A data file can be conventionally divided into two parts: 1) observed data; 2) topographic data (which is optional).

Part I of the data file: Observed data

The program accepts files with the ST extension which have the following structure.

The first line contains headers that specify the type of data in the respective columns.

The following headers for survey geometry are used:

Sx, **Sy**, **Sz** – for sources;

Rx, **Ry**, **Rz** – for receivers.

Y and Z coordinates should be present when necessary, e.g., in the following cases: surveys conducted on a grid, for 3D visualization; borehole surveys; marine (aquatic) surveys. When entering Z coordinates of sources/receivers, remember that a positive value means the distance below the surface of measurements. Negative Z coordinates are only used for marine surveys (where the surface of measurements is the bottom).

The following headers for measured values are used:

ft – first arrival times;

weight – measurement weights that determine the measurement quality. The measurement weight values should be set in the range from 0 to 1. If there is no information about the measurement errors (i.e. no column with the **weight** header in the data file), the program will automatically assign weight 1 to each measurement.

The second and subsequent lines in the data file contain the actual data corresponding to each measurement, in the same order as the headers in the first line.

If necessary, additional horizontal nodes (data points) can be entered to extend the model area beyond the ends of the array, e.g., when steep topography is present outside the array limits. The X coordinate of each node is entered in a new line after the *** symbol.

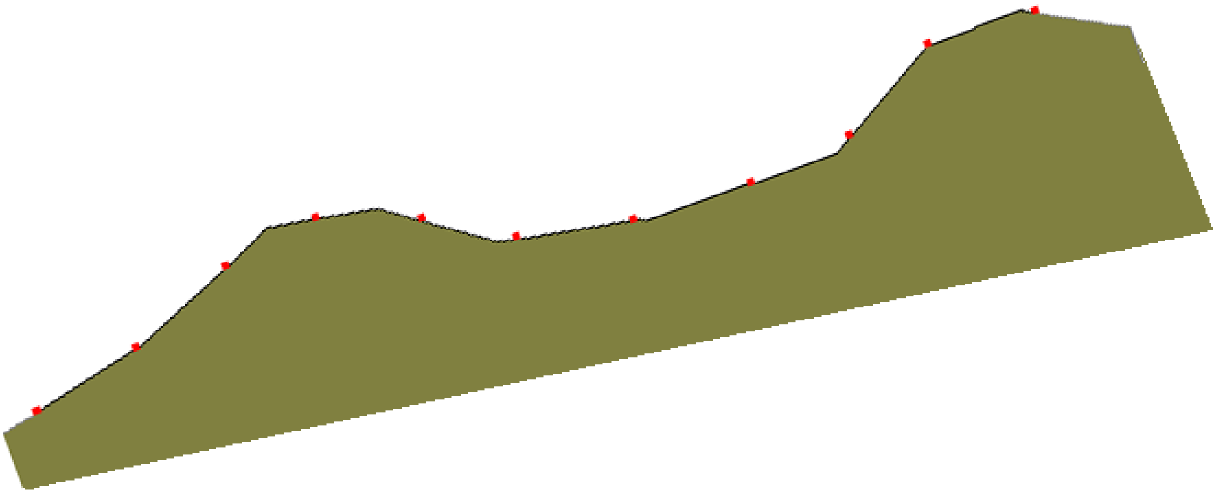
Part II of the data file: Topography data

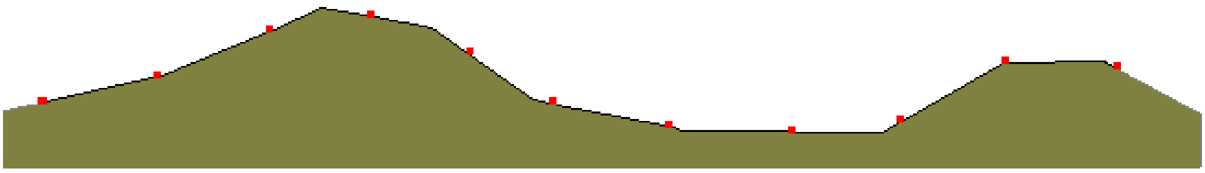
If topography data are available, a new line containing the **topo** key is started, followed by a table of coordinates and elevations.

The X coordinates of a survey line can be specified in two ways – as distances along the line and as horizontal projections. The following **topo** key ending symbols corresponding to different methods of assigning the X-coordinate of a topographic profile can be used:

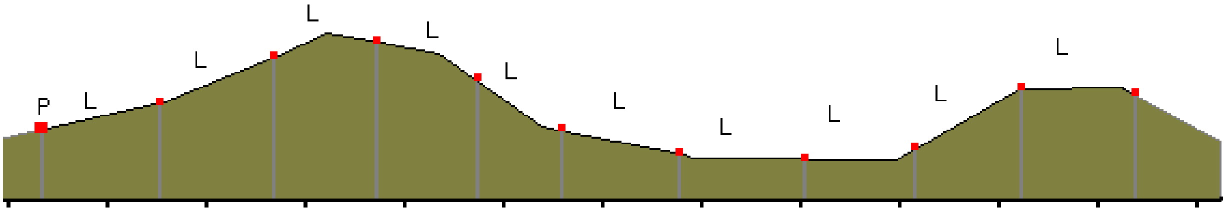
topo (no ending symbol) – receiver/source and topography coordinates are given in horizontal projections.

topo~ – reduction to a horizontal plane. The topography profile is approximated by a straight line using the least-squares method, then rotated until the line is horizontal (see figure below). This method should be applied when the survey is carried out along a slope with known elevations.





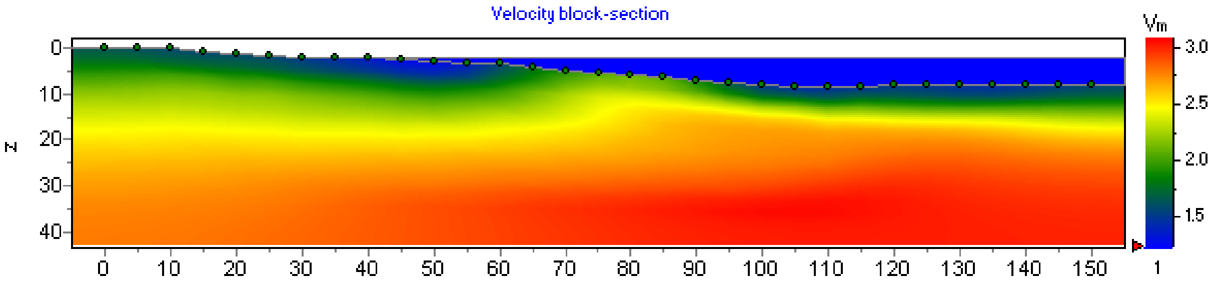
topo# – mixed method when the receiver/source X coordinates are given in line distances, topography coordinates – in horizontal projections. The horizontal coordinate of a receiver/source in this case is the distance (L) along the spread (ground length) (see figure below). In the program, the receiver coordinates are recalculated from distances to horizontal projections. It is necessary to reference one of the receivers/sources (P) to a specific topographic point. For this, following the **topo#** key, there should be a line containing the X coordinate of the topography profile point and coordinate of the corresponding receiver/source (line distance), separated by space. Then follows the table of elevations.



topo^ – this key ending is used when receiver/source and topography coordinates are given as distances along the survey line.

The above key endings can be combined, e.g. **topo~#**.

topow – this ending is used for interpretation of marine surveys (bottom or surface arrays). In this case, the topography is represented by the bottom profile or the bottom transitioning to the ground surface profile if a combination of marine and ground surveys is carried out (see figure below). In the same line, after the key, it is necessary to specify the elevation of the water surface, velocity in water and number of additional water layer partitions (3-10), separated by space (e.g. **topow** 0 1.45 5). The last two parameters can be changed in the **Options** / **Inversion** / **Underwater options** menu subsection. Then follows the table of elevations. The elevations are relative to the water surface elevation, so negative values indicate bottom topography, positive values (if any) – ground surface topography. It is possible to create combined systems, in which sources/receivers are located on the bottom and on the water surface. For this purpose, it is necessary to specify the vertical coordinates of receivers/sources (**Sz**, **Rz**) relative to the bottom elevations.



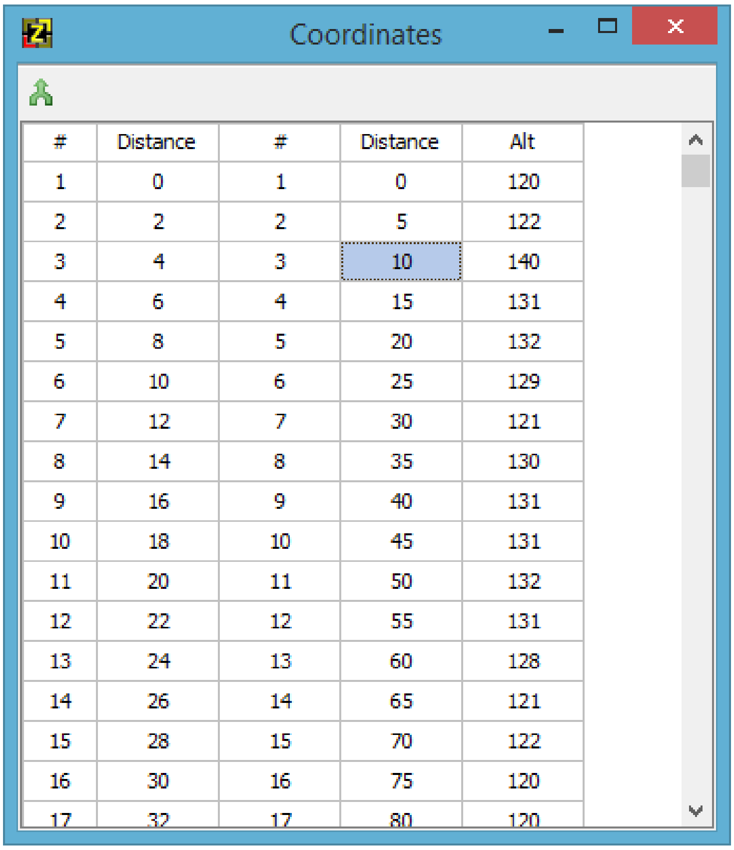
topo* – for marine surveys this ending simplifies topography data entry in the case when the spread is on the water surface (towed array).

Topography input and editing

There are several ways to input and edit topography data in ZondST2D:

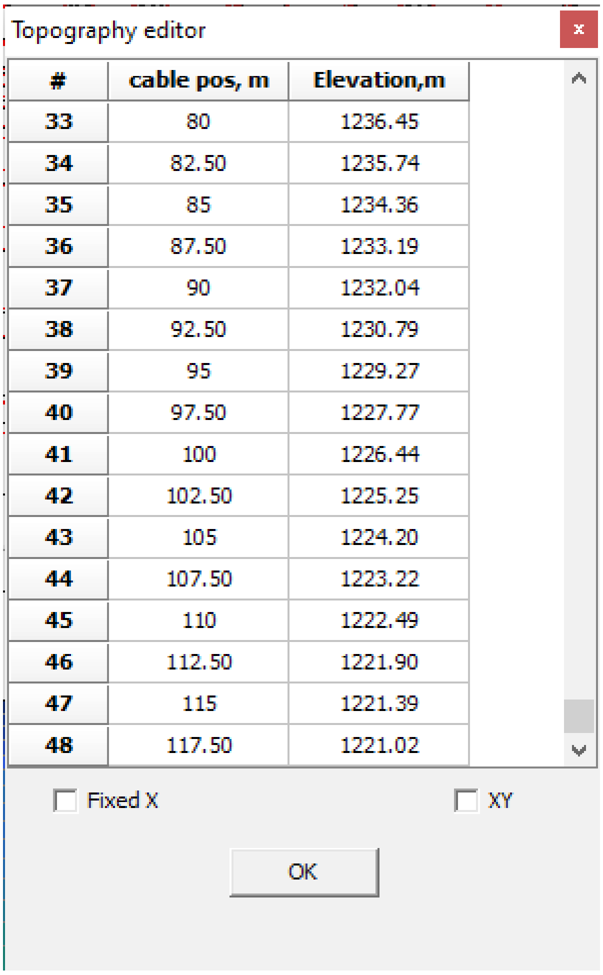
- in the **Trace editor** module at the first arrival picking stage;
- via an ST data file (for more information, see the previous section);
- by importing the data from a text file using the **Options / Topography / Import topography** menu command;
- by using **Topography editor** (**Options / Topography / Edit topography**);
- manually with the mouse using the **Options / Topography / Set by mouse** menu command.

The **Options / Topography / Import topography** command prompts to select a text file containing the topography data in two columns: distances along the survey line in one column and elevations in the other. After selecting the file, the data import window appears (see figure below). The cells in the uppermost row of the table contain a drop-down list of headers: **Distance** (distance along the survey line), **X** (projected horizontal coordinate) and **Alt** (elevation). For each column of the imported table, it is necessary to specify the corresponding header. To import the data into the project, press the  button.



#	Distance	#	Distance	Alt
1	0	1	0	120
2	2	2	5	122
3	4	3	10	140
4	6	4	15	131
5	8	5	20	132
6	10	6	25	129
7	12	7	30	121
8	14	8	35	130
9	16	9	40	131
10	18	10	45	131
11	20	11	50	132
12	22	12	55	131
13	24	13	60	128
14	26	14	65	121
15	28	15	70	122
16	30	16	75	120
17	32	17	80	120

The **Options / Topography / Edit topography** command opens the topography editor that allows you to edit the topography contained in the opened data file or imported into the project.



Topography editor

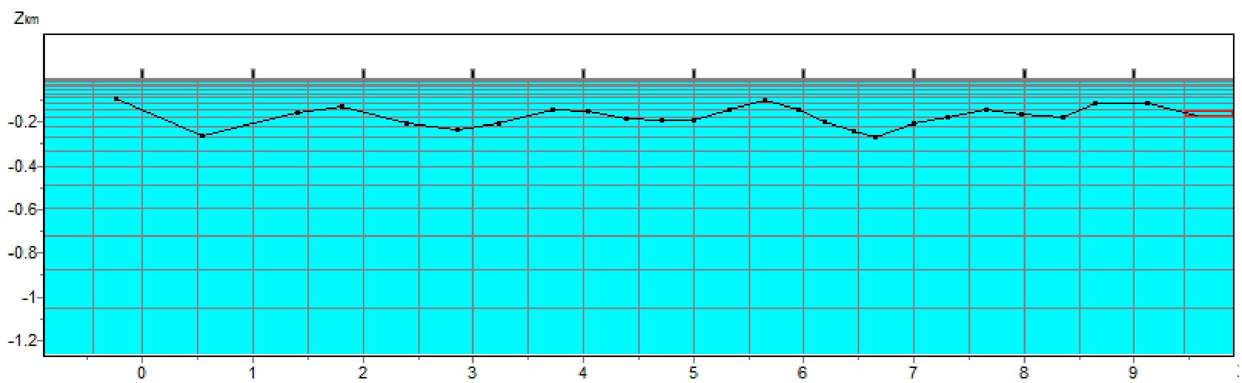
#	cable pos, m	Elevation,m
33	80	1236.45
34	82.50	1235.74
35	85	1234.36
36	87.50	1233.19
37	90	1232.04
38	92.50	1230.79
39	95	1229.27
40	97.50	1227.77
41	100	1226.44
42	102.50	1225.25
43	105	1224.20
44	107.50	1223.22
45	110	1222.49
46	112.50	1221.90
47	115	1221.39
48	117.50	1221.02

The Ctrl+X combination clears the **Elevation** column. You can then copy the data from a spreadsheet (e.g., Excel) and paste it into this column using the Ctrl+V combination.

If the **Fixed X** checkbox is checked, the X coordinates (**Cable pos**) will not change after the input of elevations; if unchecked, the X coordinates will be recalculated to projected coordinates.

The **XY** option allows adding XY coordinates (e.g., in the UTM projection) which are used when displaying the cross-section in 3D.

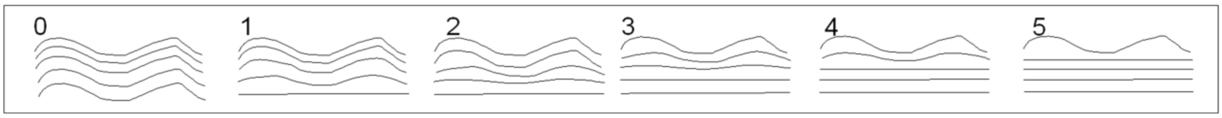
The manual topography input mode (**Options / Topography / Set by mouse**) can be useful for test purposes (generally, it is too coarse). The process of adding the topography profile is similar to adding a priori boundaries.



The coefficient of topography distortion with depth (values from 0 to 5) can be set using the **Option / Topography / Topo coefficient** menu command. If set to 0, the profile of each subsequent mesh layer will have the same geometry as the first one. If set to 1, the topography flattens with depth, and the last mesh layer will be flat (see figure below). The distorted depth is calculated using the following formula:

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

where *Topo* is elevation, and *z* is depth below the ground surface.

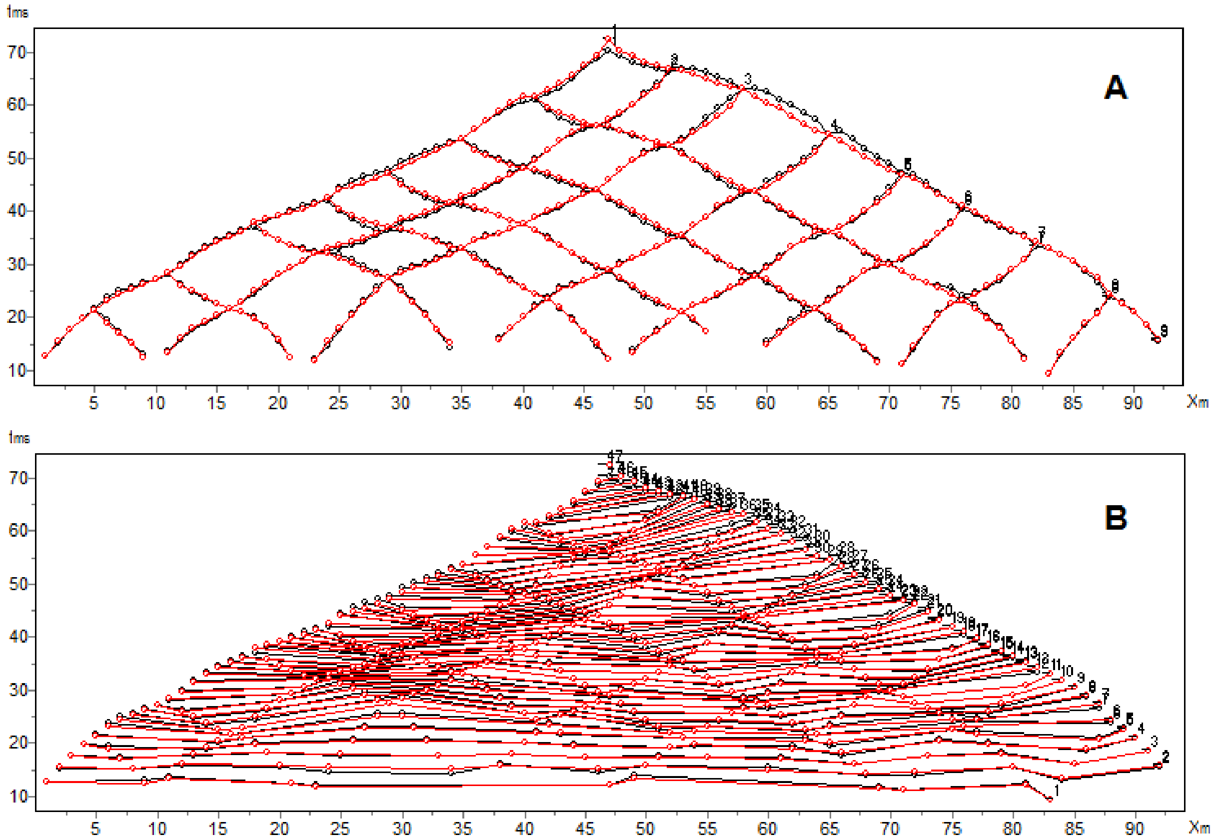


Data visualization

Graphic plot

The graphic plot displays the arrival time (**Options / Data / First break times**) or apparent velocity (**Options / Data / Apparent velocity**) values along the survey line in the form of graphs. In the attenuation mode, the amplitude values are displayed.

In the **Options** / **Data** menu subsection you can choose to plot either **Travel time curves** (example A in the figure below) or graphs for a specific source-receiver offset (**Iso S-R spacing**) (example B in the figure below).



Zooming in on a portion of the graphic area is done by pressing and dragging the mouse. To zoom in, press the left mouse button and drag the mouse up/down and to the right while holding down the button. To zoom out and return to the original scale, press the left mouse button and drag the mouse up/down and to the left. Moving (scrolling) the graphic area is done by moving the mouse with the right button pressed.

Pressing the left mouse button with the cursor pointed at a graph point clears the other graphs and displays the source and receiver positions for the selected point until the mouse button is released.

To display only one graph and hide the rest, left-click on the specific graph's index in the legend while holding down the Shift key. If you click again, the operation is reversed. To scroll through the graphic plot, select one or several adjacent graphs in the legend and spin the mouse wheel, with the mouse pointer hovering over the legend. The indexes of the active graphs will change.

If the **Data editor** window is opened (**Options / Data / Data editor**), right-click on a graph point will highlight the corresponding measurement in the table.

Graph properties such as line thickness, color, etc. can be changed in the **Graphics Setup** window accessible with the **Options** / **Graphics** / **Observed graphics** or **Calculated graphics** commands.

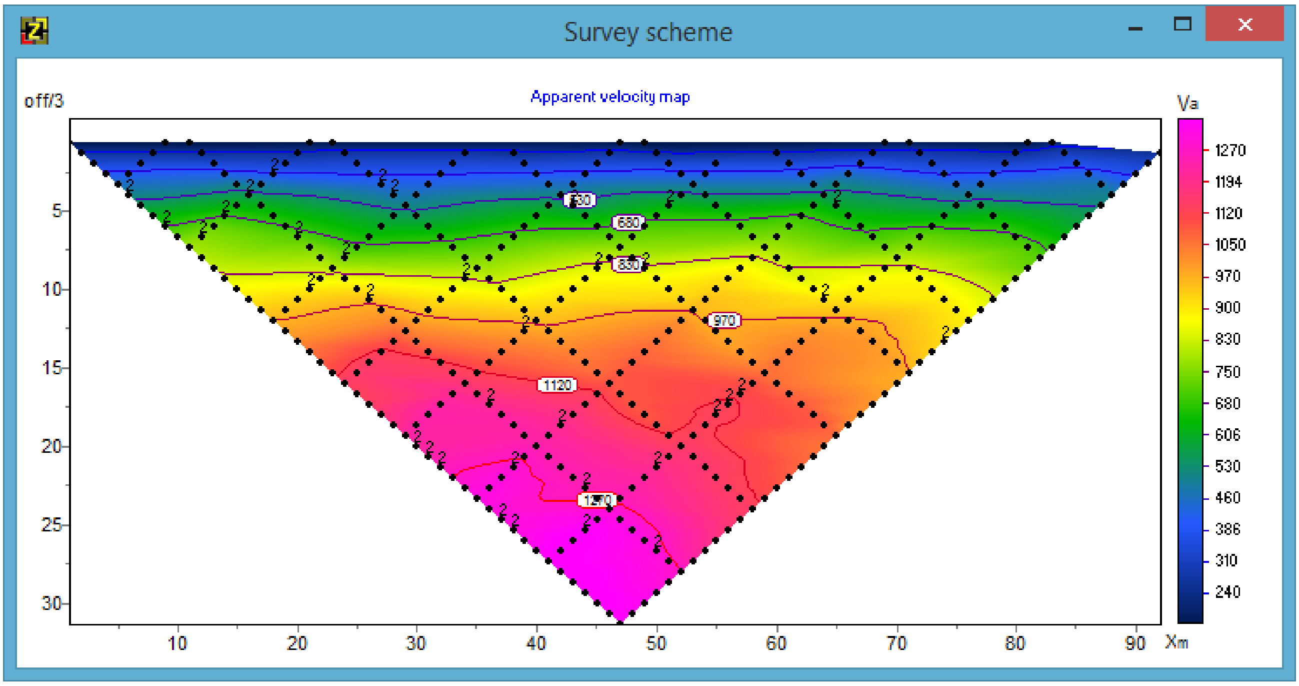
Values of individual data points can be edited by pressing the left mouse button and dragging a graph point up/down while holding down the Ctrl key. To delete a data point, click the middle button of the mouse (wheel) while holding down the Alt key. Deleting is performed in a circular window which size can be adjusted by spinning the mouse wheel.

If measurement weights are specified in the input data file, you can display the confidence intervals on the graphs using the **Options** / **Data** / **Display error gates** command. The confidence intervals (weights) can be adjusted on the graphic plot by pressing the left or right mouse button while holding down the Alt key.

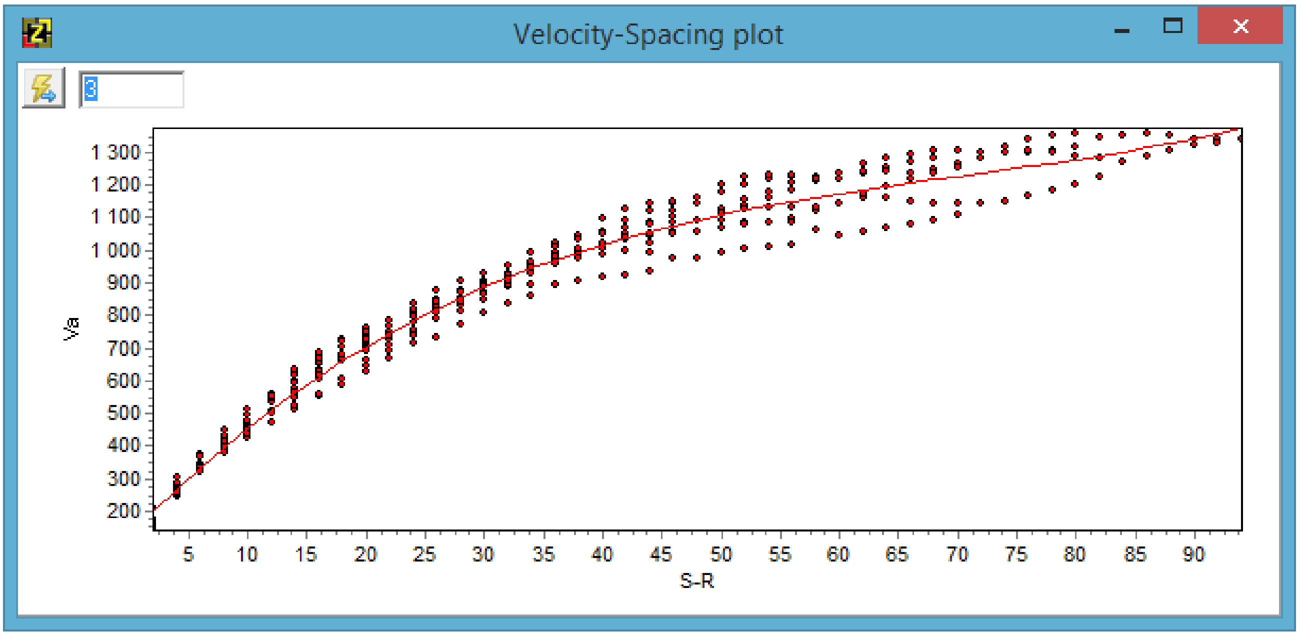
The **Axis Editor** is opened by right-clicking on the axis while holding down the Shift key (when hovering over the axis, the mouse pointer turns into a hand pointer). A context menu containing three items (**Options**, **Default**, and **Fix range**) will appear (for more information, see the **Axis editor** section).

Additional data visualization options

Additional options for measurement data visualization and analysis available in the **Options / Data** menu subsection include **Survey pseudosection** and **Velocity offset plot**. When the **Survey pseudosection** command is selected, the *Survey scheme* window containing a pseudosection of apparent velocities appears.



The **Velocity offset plot** command opens a plot of apparent velocities as a function of offset in a separate window. In this window you can set the initial model for the inversion using the button. The coefficient in the field next to the button sets the ratio between depth and offset.

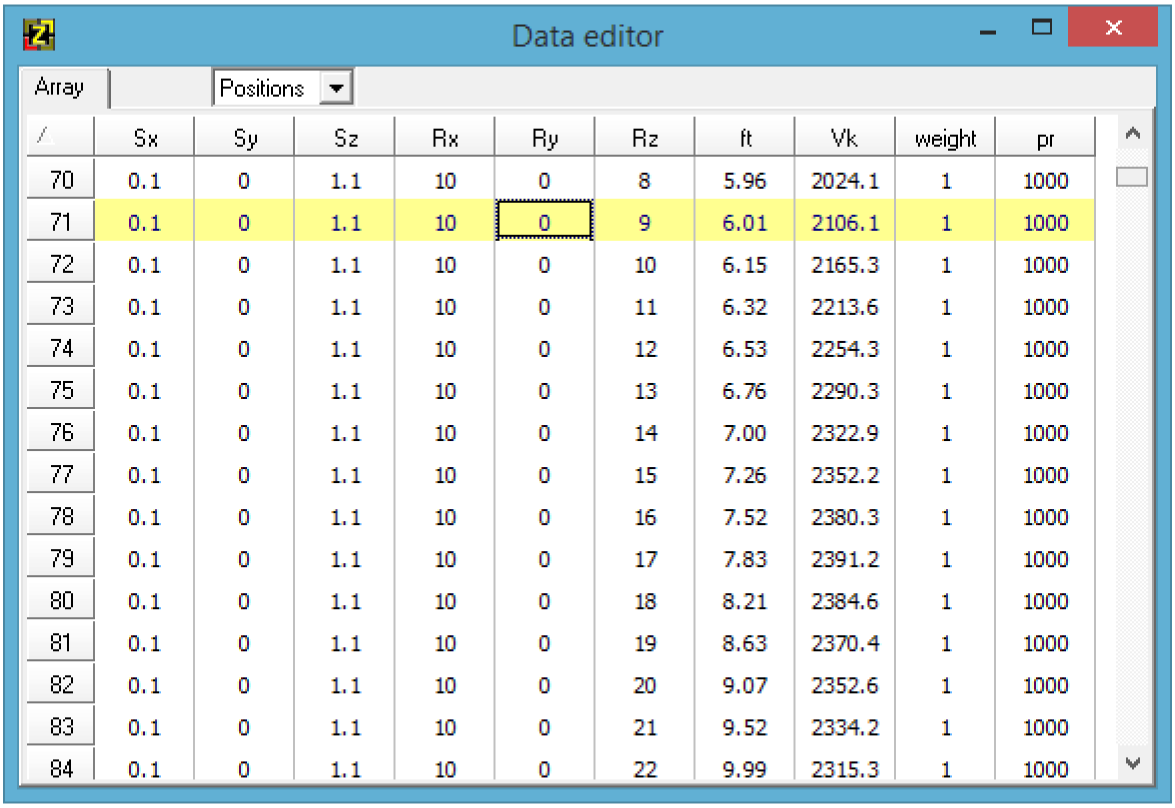


Data editor

The **Data editor**, which can be opened by pressing the button on the toolbar or selecting the **Options / Data / Data editor** menu command, is used for viewing the observation parameters and measured data. The data editor window contains a table consisting of ten columns:

Sx	Source X position*.
Sy	Source Y position.
Sz	Source Z position (depth).
Rx	Receiver X position*.
Ry	Receiver Y position.
Rz	Receiver Z position (depth).
ft	First arrival time.
Vk	Apparent velocity.
Weight	Measurement weight.
pr	Survey line ID.

* By switching between the **Positions** and **Coordinates** options above the table, the X of sources and receivers can be displayed in the table as initial coordinates (distances) as recorded in the data file (when the **Coordinates** option is selected), or as calculated horizontal projections (when the **Positions** option is selected).



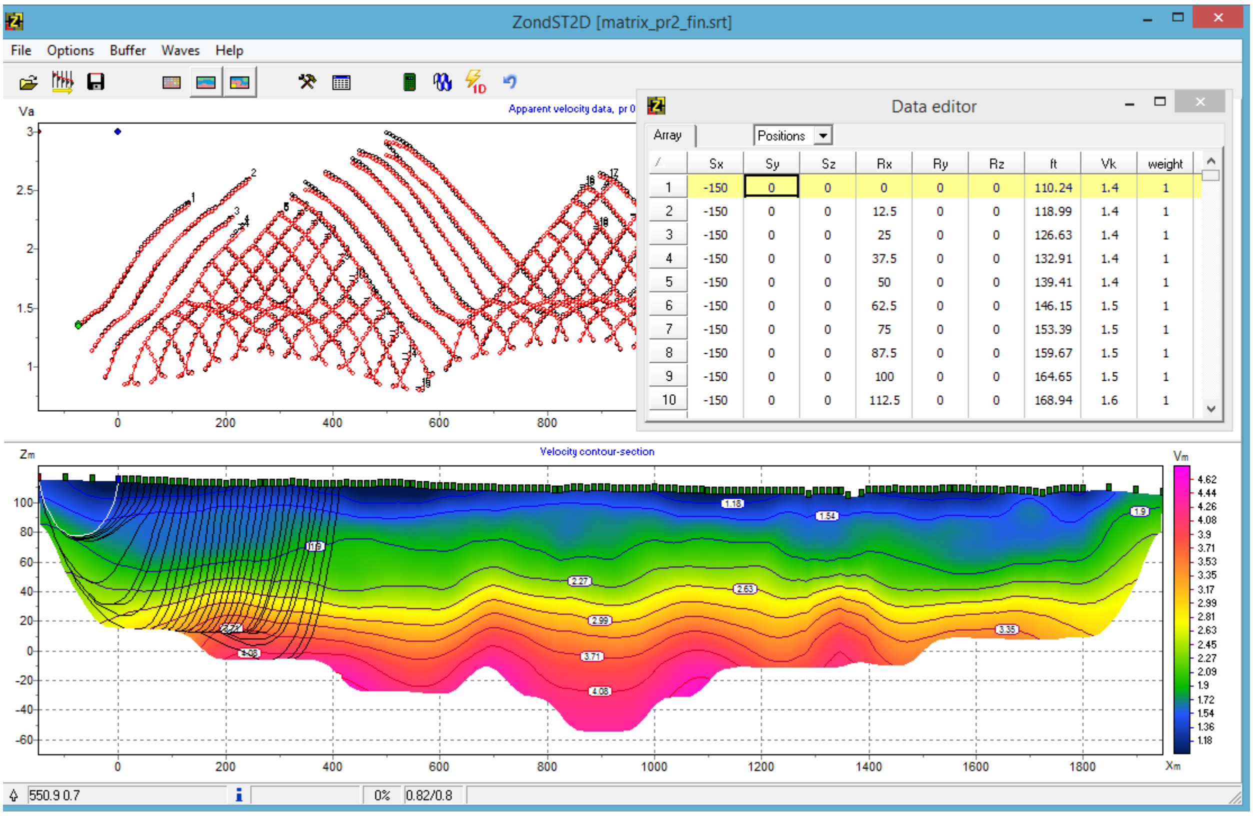
Data editor

Array | Positions

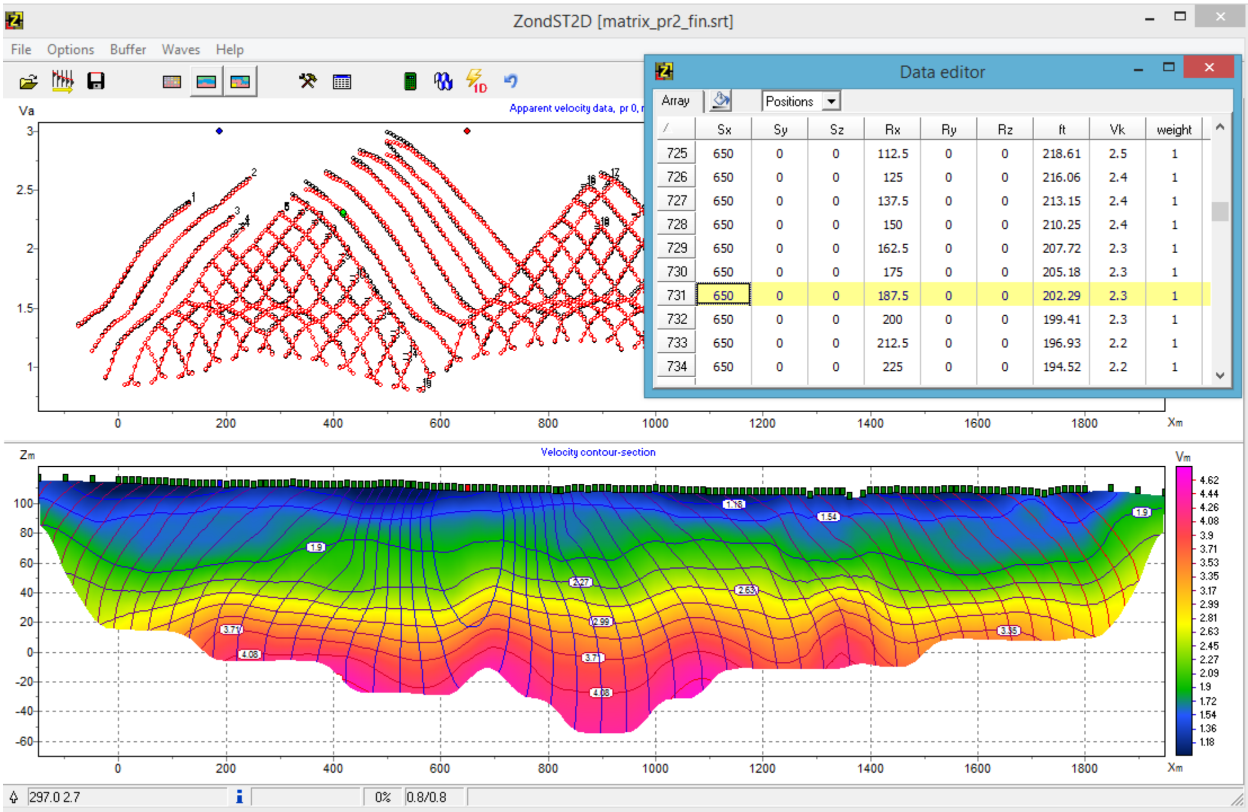
/	Sx	Sy	Sz	Rx	Ry	Rz	ft	Vk	weight	pr
70	0.1	0	1.1	10	0	8	5.96	2024.1	1	1000
71	0.1	0	1.1	10	0	9	6.01	2106.1	1	1000
72	0.1	0	1.1	10	0	10	6.15	2165.3	1	1000
73	0.1	0	1.1	10	0	11	6.32	2213.6	1	1000
74	0.1	0	1.1	10	0	12	6.53	2254.3	1	1000
75	0.1	0	1.1	10	0	13	6.76	2290.3	1	1000
76	0.1	0	1.1	10	0	14	7.00	2322.9	1	1000
77	0.1	0	1.1	10	0	15	7.26	2352.2	1	1000
78	0.1	0	1.1	10	0	16	7.52	2380.3	1	1000
79	0.1	0	1.1	10	0	17	7.83	2391.2	1	1000
80	0.1	0	1.1	10	0	18	8.21	2384.6	1	1000
81	0.1	0	1.1	10	0	19	8.63	2370.4	1	1000
82	0.1	0	1.1	10	0	20	9.07	2352.6	1	1000
83	0.1	0	1.1	10	0	21	9.52	2334.2	1	1000
84	0.1	0	1.1	10	0	22	9.99	2315.3	1	1000

When you select a data point in the table, the corresponding source and receiver positions of the data point are indicated on the graphic plot and in the model section.

When the **Options / Data / Ray paths** option is enabled, the ray paths in the model section are displayed only for the source position selected in the table (see figure below).



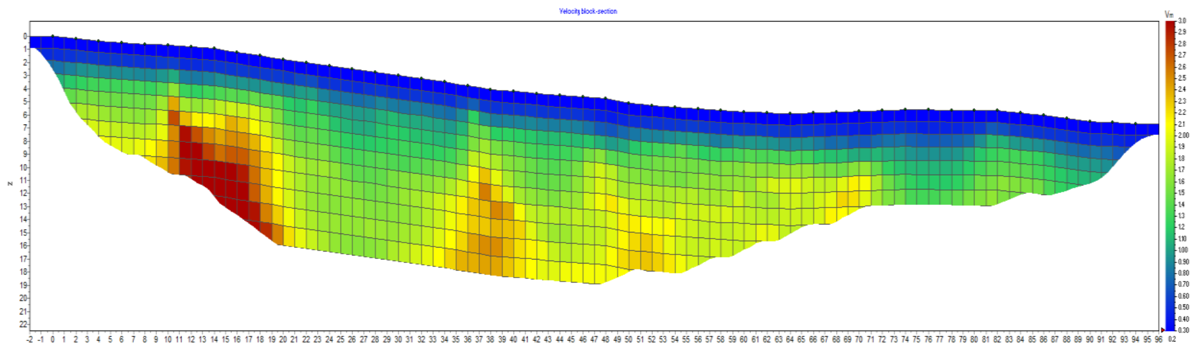
The **Data editor** is also displayed when selecting the **Options / Extra / Display t0 map** menu command. After the calculation is completed, the delay time (t0) isolines map for the source position selected in the table will be displayed in the model section. The isolines can be configured using the button on the toolbar of the data editor window.



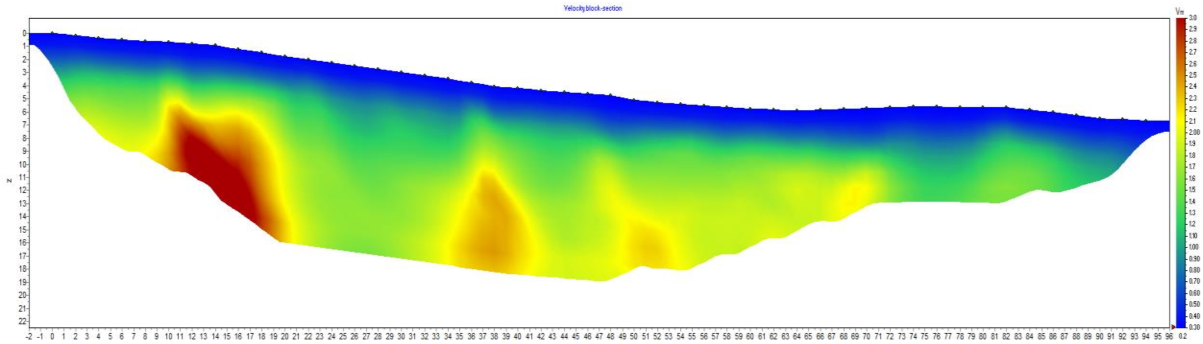
Visualizing the model

In the tomography (mesh) modeling mode, the model section can be displayed in the form of blocks (cells), in the smooth interpolated palette mode, or in the form of color contours (in the figure below, sections A, B and C, respectively). The **Options / Model / Block-section**, **Smooth-section** and **Contour-section** commands are used to switch between the visualizing modes.

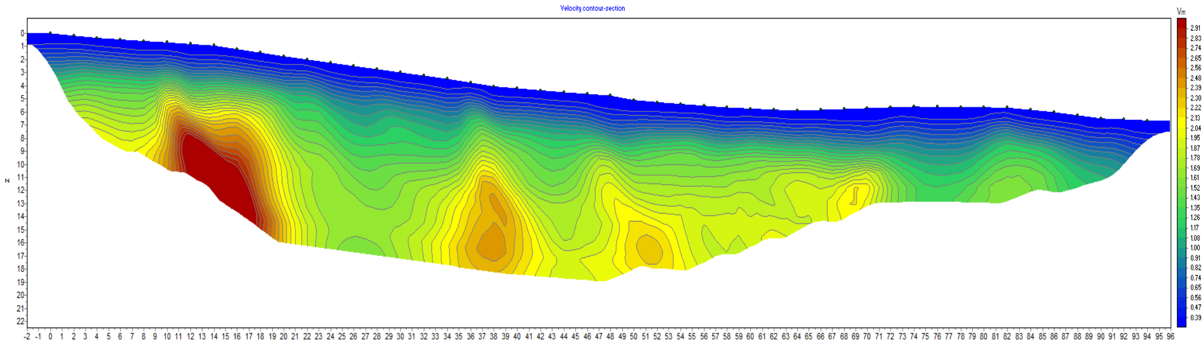
A



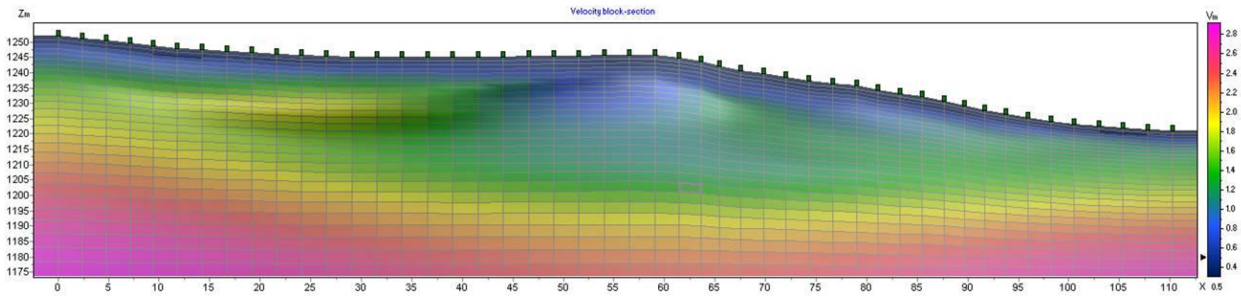
B



C



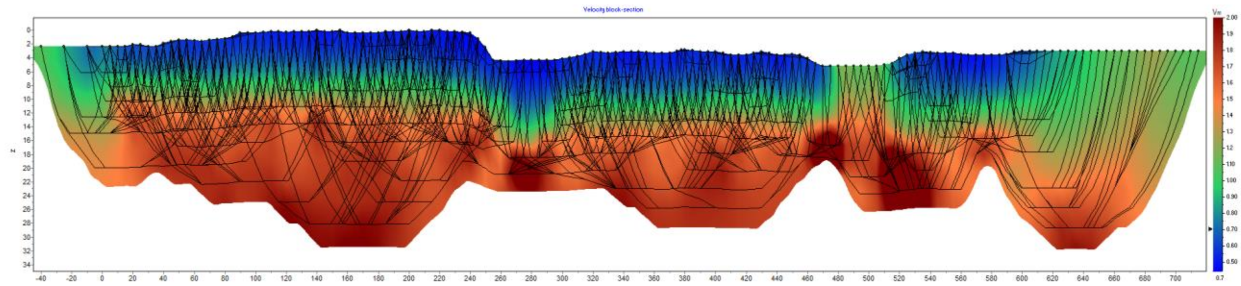
The **Options / Import/Export / Background image / Create shaded map** option allows showing a shaded relief map based on the current model in the background.



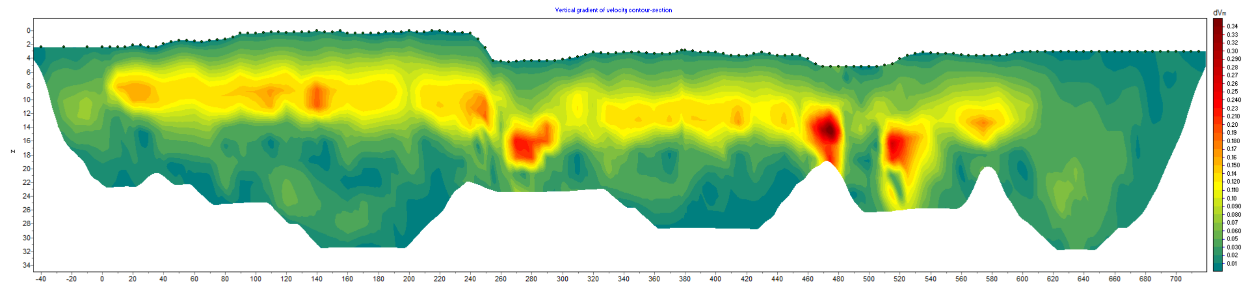
Some of the program options and features depend on the chosen mode of model display. Mathematical modeling should be performed in the **Block-section** mode. In the **Contour-section** mode, you can calculate and display the distribution of sensitivity (**Option / Model / Sensitivity**) and full velocity gradient (**Option / Model / dVelocity**) (in the figure below, sections C and B, respectively).

Reliability of modeled velocity distribution for a certain portion of the model is determined by the density of ray paths. The **Option / Data / Ray paths** command allows displaying the ray path scheme calculated for the current model during the inversion. The **Options / Model / Cut by rays** command allows hiding the portions of the model not covered with the ray paths (section A in the figure below).

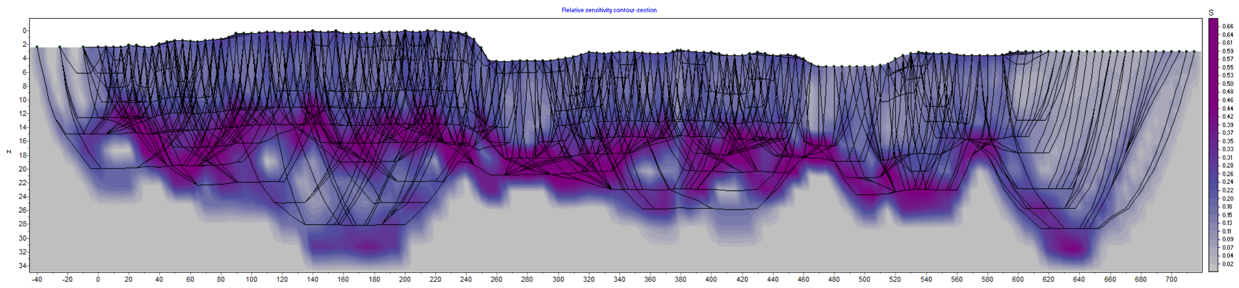
A



B



C

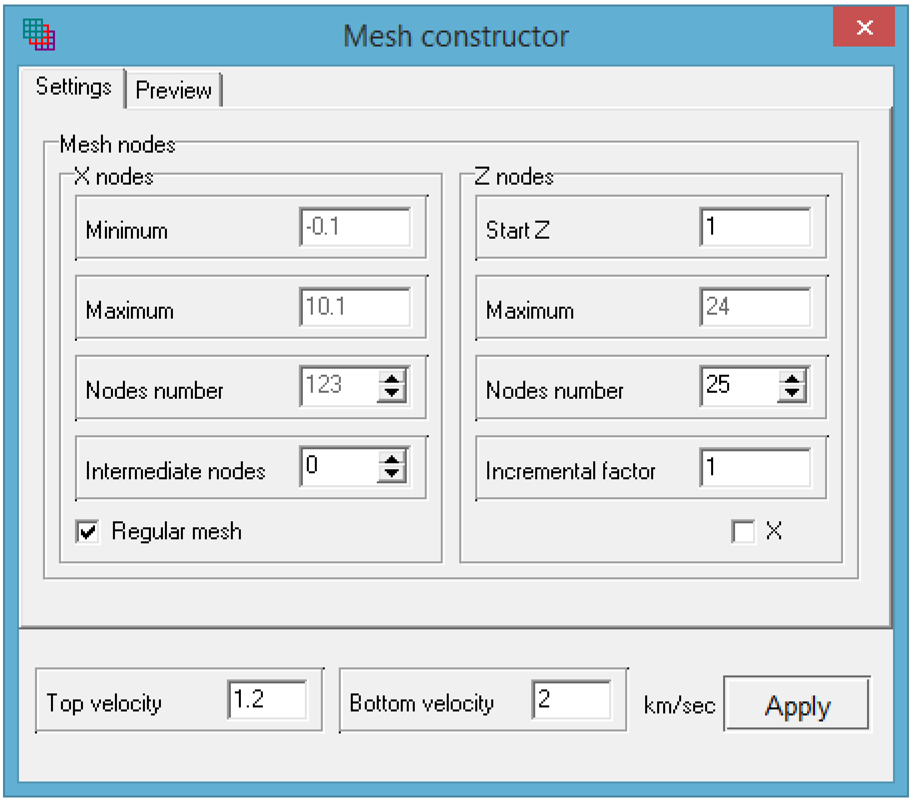


Modeling

Modeling is an important process preceding fieldwork. Using a priori information about the target object, the geophysicist can model various geological settings when planning geophysical work, which allows optimizing the fieldwork technique for solving a specific geological problem.

Creating a mesh model

After creating a new measurement system for modeling, opening a data file, or performing picking in the **Trace Editor** module, a dialog box for setting up the initial model mesh appears. To create a new model or change the parameters of an existing model, you can use the **Options / Mesh constructor** command. In the **Mesh Constructor** dialog box, you are prompted to select the mesh parameters and the initial velocity of the model.



The **X nodes** group box contains options that allow you to set the parameters of the model mesh in the horizontal direction.

Minimum – specifies the minimum horizontal coordinate of the survey line.

Maximum – specifies the maximum horizontal coordinate of the survey line.

Nodes number – specifies the number of horizontal model nodes. If the interpretation is performed for a downhole survey (i.e., there is only one X coordinate for the whole data set), this option specifies the number of additional nodes to the left and to the right of the borehole.

Intermediate nodes – sets the number (0-4) of additional nodes between unique source/receiver positions on the survey line. If the interpretation is performed for a cross-hole survey (i.e., there are only two X coordinates for the whole data set), a larger number of intermediate nodes should be specified or the **Regular mesh** option should be enabled.

Regular mesh – activates the mesh constructing algorithm in which additional nodes are created based on the condition of the uniform subdivision. The option should be enabled in the case when the spacing between adjacent sources/receivers varies significantly. By right-clicking on the area with the **Regular mesh** label you can specify the step of cell subdivision in the horizontal direction (the **Nodes number** option should be disabled).

The **Z nodes** group box contains options for defining the model mesh parameters in the vertical direction. Initially, the program automatically selects these parameters in accordance with the following rules:

- The depth to the deepest layer equals a third of the maximum spread length.
- The number of layers roughly corresponds to the number of unique source-receiver offsets for a given measurement system.
- The thickness of each consequent layer is 1.05 times the thickness of the previous layer.

But for many types of surveys, especially for cross-hole surveys, the parameters should be specified manually.

Start Z – sets the thickness of the first layer. The value should correspond approximately to the horizontal size of the cell and meet the required resolution of the survey. If the **X** checkbox is activated (available when borehole data are interpreted), the **Start Z** label is replaced by **Minimum**, and in this case, the value indicates the minimum vertical coordinate of the model, which is used when the mesh needs to be started from a certain depth.

Maximum – indicates the depth of the deepest layer. This parameter is calculated automatically. Note that for surface-based surveys, the maximum depth should not be overly deep because the penetration depth of the method is limited by the spread length and geology. For borehole surveys, the value should be greater than the source/receiver maximum depth.

Nodes number – sets the number of model layers. Typically, 15-18 layers are enough for a comprehensive model in the case of surface-based surveys. If the **X** checkbox is activated (available when borehole data are interpreted), the value specifies the number of vertical nodes of the model (determined based on the number of unique source/receiver positions in a borehole).

Incremental factor – sets the ratio between the thicknesses of adjacent layers for surface-based surveys. The values are typically chosen in the range from 1 to 2. If the **X** checkbox is activated (available when borehole data are interpreted), the value determines the number of intermediate vertical nodes in the mesh (between sources/receivers).

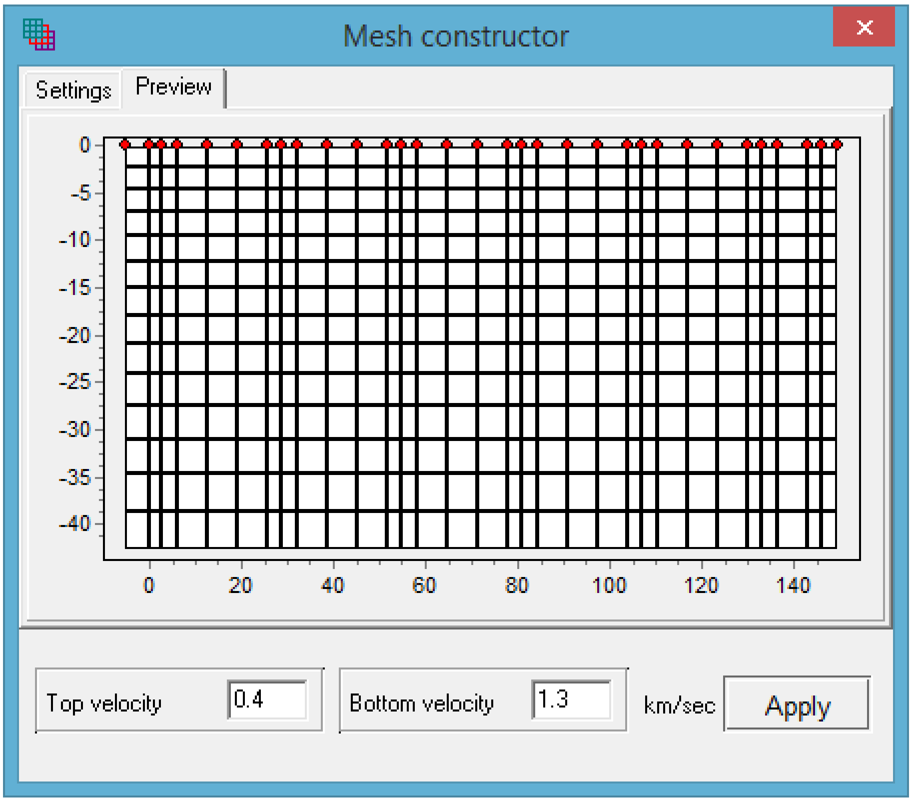
When the **X** checkbox is activated, vertical nodes of the model mesh are set to receiver positions. This option is available when interpreting borehole data.

Regular mesh (available when working with borehole data) – includes an algorithm for building a vertical network in which additional nodes are selected based on the condition of uniform partitioning. The option should be enabled in case of very different vertical distances between neighboring sources/receivers.

Top velocity – sets the velocity of the topmost layer of the initial model.

Bottom velocity – sets the velocity of the deepest layer of the initial model. The velocity values of intermediate layers are determined by linear interpolation between the values in the first and the last layers.

The **Preview** tab shows the preview of the configured mesh without topography.



After configuring the mesh, press **Apply** to proceed to the working mode.

You can manually edit the created mesh (add or remove intermediate mesh nodes, adjust the height and width of cells) using the **Model editor** tools (see below).

Editing the model

Velocity modeling is performed in the **Model editor** – the lower graphical section of the main program window.

ZondST2D provides the following three modeling modes:

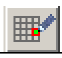
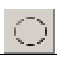
- Tomography (mesh) mode (the button on the main window toolbar) is intended for direct editing of the model cells (the width and thickness of a row or column of cells, as well as cell properties). When working in block mode, it is also possible to edit the mesh geometry initially created in the **Mesh constructor**.
- Layered mode (the button on the toolbar) is intended for creating and editing layered models.
- Polygonal mode (the button on the toolbar) is intended for creating and editing models consisting of a set of connected or disconnected objects.

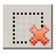
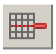
In each of the model modes, various auxiliary tools facilitating the modeling process are available, such as the import of borehole logs, background images, models from other programs of the **Zond** package.

Tomography (mesh) model mode

This mode is used to interactively change parameters of individual model cells or groups of cells. Working with the model cells is similar to editing a raster image in graphical editors. The active cell in the model is highlighted with a rectangle – cursor. When you move the cursor in the model area, the status bar at the bottom of the main program window displays the coordinates and parameters of the cell in which the cursor is located. A selected or fixed cell is indicated with a hatch of white or black dots, respectively.

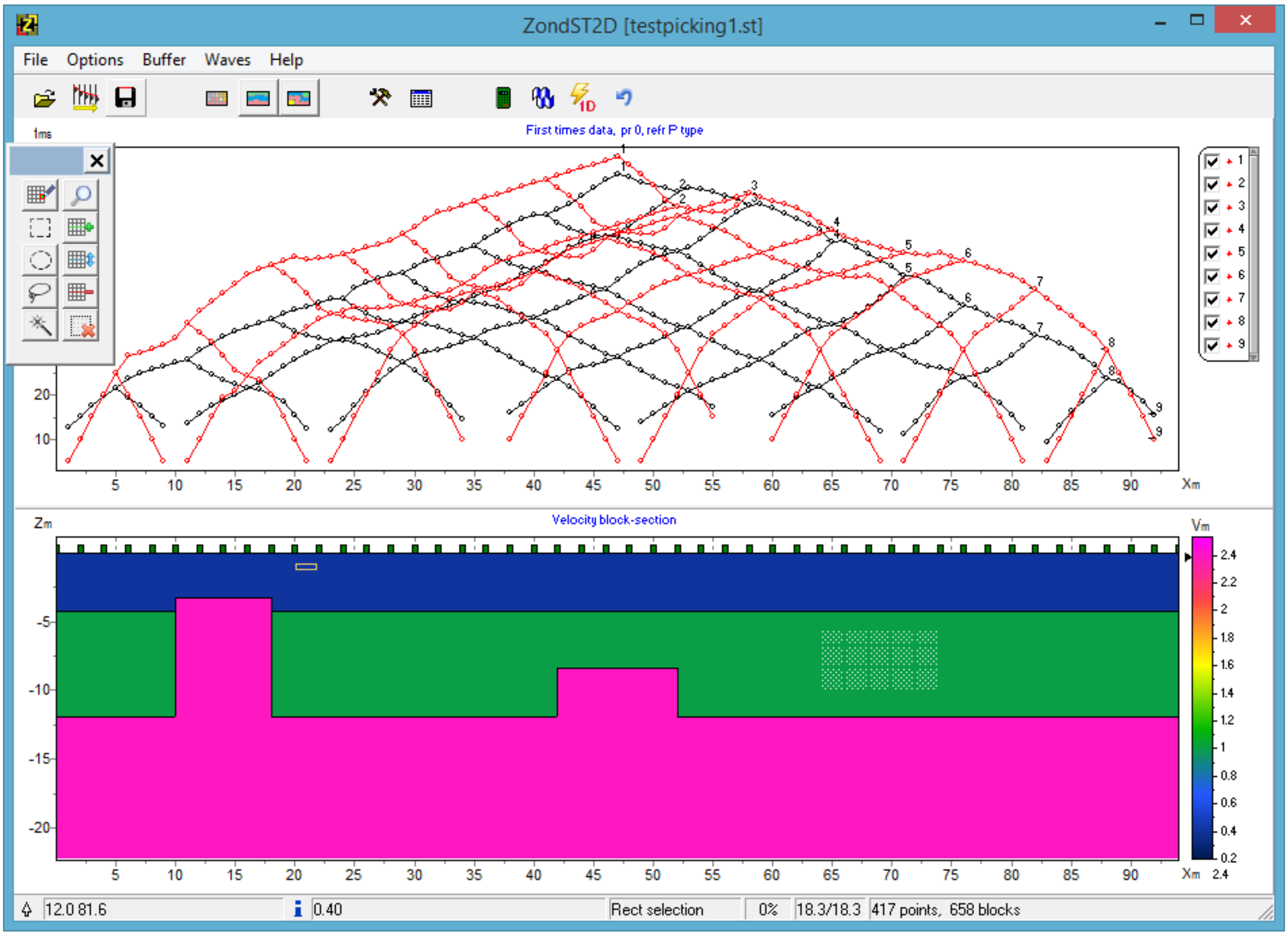
There are several sets of tools for working with the mesh model available in the program. The two main sets of tools are accessible through the context menu (brought up by right-clicking in the model area) and the floating toolbar (opened with the **Options** / **Model** / **Model editor toolbar** command). The items in the context menu and the floating toolbar generally duplicate each other.

Context menu item	*Toolbar item*	*Action*
		Zoom in by pressing the left mouse button and dragging the mouse up /down and to the right. To zoom out, drag the mouse in the opposite direction.
Display cell setup		Open the **Cell Setup** dialog box.
Cell to cursor value		Use the parameter of the active cell as the current value.
Edit mode		Enable editing mode.
Selection / Free form selection		Select an arbitrary group of cells.
Selection / Rectangular selection		Select a rectangular group of cells.
		Select an elliptical group of cells.
Selection / Magic wand		Select a group of cells based on the similarity of the cells' parameter values. The active cell and the adjacent cells with similar parameter values are selected. The degree of similarity is set in the **Model setup** dialog box (right-click on the color scale or the title of the model section and select **Setup**). This action can also be performed by pressing the X key.

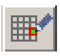
Selection / Remove selection		Remove selection.
Mesh options / Add column/row		Add a new vertical or horizontal mesh boundary. The new boundary appears when you click on the desired location.
Mesh options / Remove column/row		Remove a mesh boundary by clicking on the boundary.
Mesh options / Resize column/row		Change the thickness of a row or column using the mouse.
Clear model		Clear the model completely. Remove all selections and reset the model to the initial settings specified in the **Mesh constructor**.
Clear parameters		Reset the model to the initial settings specified in the **Mesh constructor**. The selections made and the parameter limits specified by the user are retained.

The color scale to the right of the model section indicates the colors assigned to specific data values. To select a current value, left-click on the scale; the value is displayed below the color scale. To assign the current value to a cell, left-click on the cell.

The program provides several methods of cell selection – rectangle, ellipse, free-form and by a certain parameter value (the “Magic wand” tool), accessible through the context menu or floating toolbar. If some area of the model section is selected, selecting the current value on the color scale and then left-clicking on any place in the selected area sets all cells of the area to the current value. For example, to give a rectangular area of the model section a certain velocity value, select the desired area using the rectangular selection tool, then left-click on the color scale at the desired velocity value, and left-click on any cell of the selected area of the model.



The **Zoom&Scroll** mode is enabled by pressing the Ctrl+A combination, clicking the  button on the floating toolbar or by selecting the **Zoom&Scroll** option in the context menu brought up by right-clicking on the **Velocity block-section** title of the model section. To zoom in on a portion of the model area, press the left mouse button and drag the mouse up/down and to the right. To zoom out, drag the mouse in the opposite direction.

In the **Edit mode** (activated with the  button on the floating toolbar or by selecting the **Edit mode** option in the context menu brought up by right-clicking in the model area), left-clicking on a cell or selection while holding down the Shift key increases its parameter. Right-clicking on a cell or selection while holding down the Shift key decreases its parameter. The percentage by which the value is changed is specified in the **Model Setup** dialog box (see the **Model section settings** section).

Pressing and holding down a mouse button while holding down the Ctrl key allows you to move a selected group of cells within the editing area. With the left mouse button pressed, the contents of the selection are copied to a new location. With the right mouse button pressed, the contents of the selection are cut and copied to a new location.

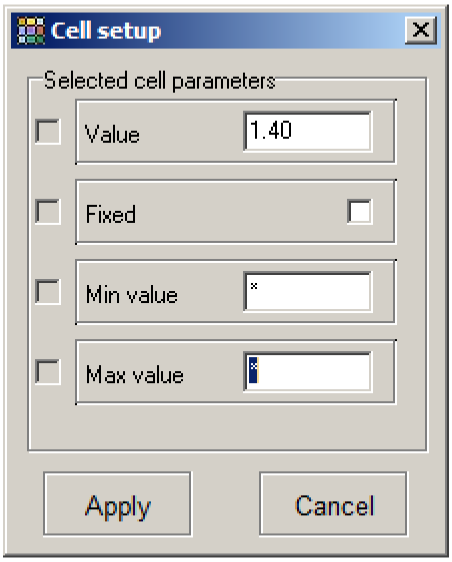
Parameter values for selected cells can be set or edited using the **Cell setup** dialog box (right-click on a cell or group of cells – **Display cell setup**). The following options are available:

Value – sets the value of the cell parameter.

Fixed – fixes the cell parameter.

Min value, **Max value** – defines the range of variation of the cell parameter for inversion.

If the checkboxes to the left of the option boxes are checked, the changes will be applied to all selected cells.



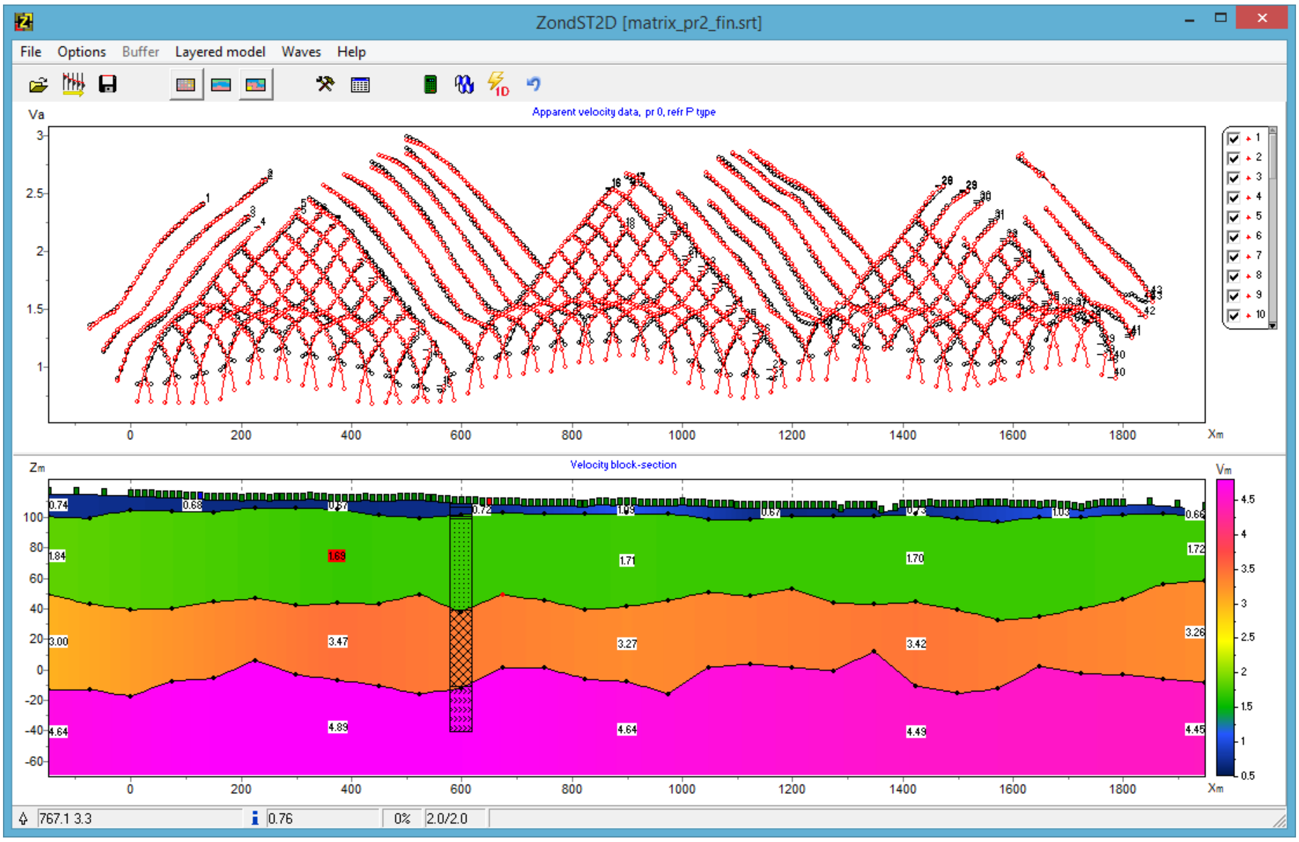
Right-clicking in various areas of the model editor will bring up context menus with the following options:

Area	*Option*	*Description*
Vertical axis	Set maximum	Set the depth of the bottom layer.
	Redivide	Set the same layer thickness for all model layers.
	Thick mesh	Remove every other node in the vertical direction.
	Thin mesh	Add intermediate nodes in the vertical direction.
Horizontal axis	Redivide	Set the same width for cells located between unique source/receiver positions.
	Thick mesh	Remove every other node in the horizontal direction (only removes a node if there is no electrode located in this node).
	Thin mesh	Add intermediate nodes in the horizontal direction.
Model title area	Display model mesh	Show/hide the model mesh.

	Display objects border	Show/hide the object boundaries.
	Setup	Open the **Model Setup** dialog box.
	Display color bar	Show/hide the color scale.
	Zoom&Scroll (Ctrl+A)	Enable the **Zoom&Scroll** mode.
	Print preview	Print the model.
Color scale	Setup	Open the **Model Setup** dialog box.
	Set range	Set the minimum and maximum values of the color scale manually.
	Automatic	Automatically determine the minimum and maximum values of the color scale.
	Log scale	Set the logarithmic scale for the color scale.
	Set half-space value	Set the parameter value of the half-space (the background value).
	Set cursor value	Set the current value of the parameter.
	Colors as histogram	Set the colors based on the distribution of velocity in model cells.

Layered model mode

A layered model consists of a set of layers with arbitrary geometry and velocity variations of each layer. It is similar to the block (mesh) mode, in which the medium is divided into cells, but in a layered representation. This representation is more structured and often the most reasonable from a geological perspective. Considerations of velocity changes within a layer are especially important for geotechnical surveys, where lateral velocity changes are as significant as the changes with depth. **ZondST2D** implements an accurate solution of forward and inverse problems for a layered medium, allowing the wave to propagate along the shortest distance path, contrary to the approach implemented in the reciprocal method where the wave always propagates along a refractive boundary (with no restrictions on the path curvature).



When the layered mode is activated with the  button on the toolbar, a **Layered** sub-menu appears in the main window menu.

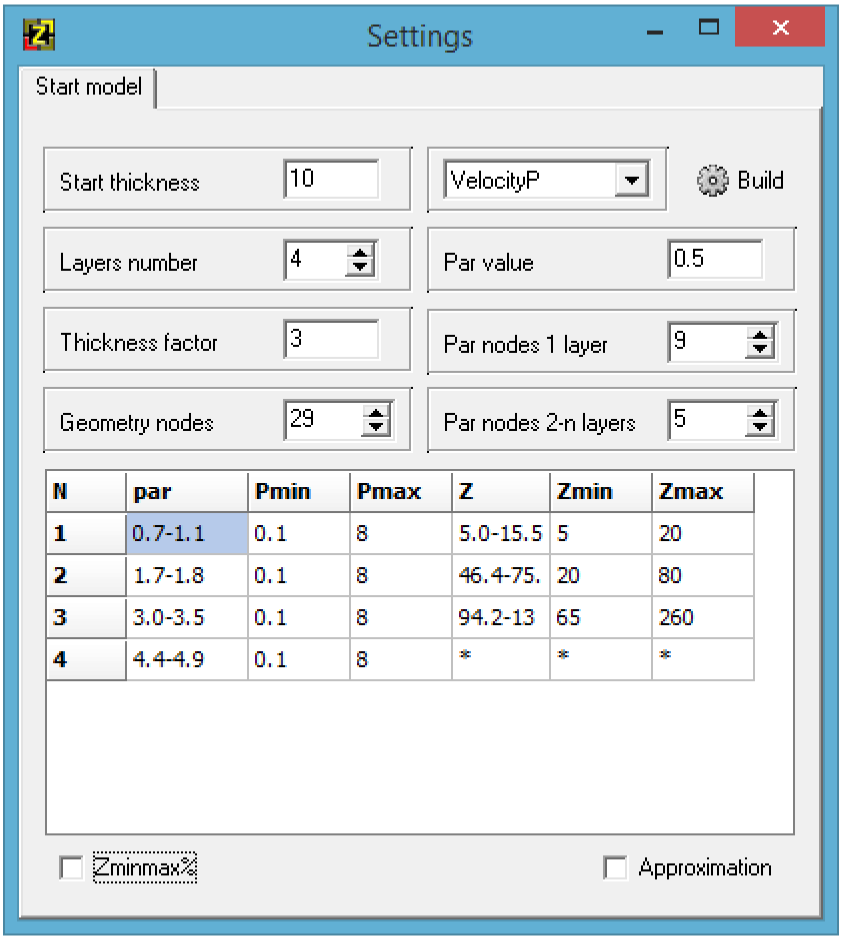
An initial model in the form of a horizontally layered medium with constant layer parameters can be set using the **Layered / Model constructor** menu command. When setting up the initial model, it is important to create a good approximation of the real geology for inversion. The initial model can be specified based on the analysis of the block (mesh) model inversion results. The results of the mesh model inversion can be incorporated into the framework of the created layered model using the **Layered / Load from mesh** menu command.

The first set of parameters defining a layered model is the number of layers and their thickness and velocity gradient (velocity increase from layer to layer). Next, the number of boundary nodes (geometry nodes) and the number of parameter variation nodes within a layer (parameter nodes) are specified. A geometry node is a point with a changeable vertical coordinate. More complex boundaries can be specified by increasing the number of geometry nodes. On the other hand, overcomplicating the geometry might decrease the reliability of the resulting model. It is recommended to choose the number of geometry nodes based on the number of unique receiver positions on the survey line. All subsurface boundaries have the same number of nodes. In the edit mode (**Layered / Edit mode**), nodes are displayed as circles that can be dragged vertically with the mouse or fixed for inversion. To fix a node, left-click on the node while holding down the Ctrl key; fixed nodes are displayed in red. It is practical to edit model geometry with the **Layered / Transparent** option enabled to display the results of inversion in mesh mode in the background.

Parameter nodes, or parameter variation profile, define the velocity (or another parameter) distribution within a layer. The parameter variation profile is specified by a set of fixed node values, with linear interpolation between the values. The program implements a different number of nodes for the topmost layer and the subsequent layers. Typically, the topmost layer is characterized by higher variability of seismic velocities and a more detailed profile, consisting of 3-10 nodes, should be specified for it. In the deeper layers, velocity changes more smoothly, and the profile can be limited to 1-3 nodes. In the edit mode (**Layered / Edit mode**), you can edit node parameter values by right-clicking on the value label. To fix the parameter value for inversion, right-click on the value label while holding down the Ctrl key.

Layered models can be used to perform a joint inversion with data of other geophysical methods to obtain a model with common geometry of boundaries and positions of parameter nodes. Joint inversion is possible between the following data types: seismic refraction (V_P), seismic refraction (V_S), seismic reflection, MASW, VES, gravity, magnetic, MT and TEM. Before conducting a joint inversion, it is necessary to specify each method's model individually.

An initial layered model can be configured in the **Layered / Model constructor** window.



Start thickness – sets the thickness of the first layer.

Layers number – sets the number of layers.

Thickness factor – sets the thickness increment for each successive layer.

Geometry nodes – sets the number of geometry nodes for each boundary (from 10 to 30).

– sets the type of parameter from the following list:

Gravity – density for gravity data;

Magnetic – magnetic susceptibility for mag data;

VelocityP – P-wave velocity for seismic refraction or reflection data.

VelocityS – S-wave velocity for seismic refraction or MASW data.

Resistivity – resistivity for VES, MT or TEM data;

If the selected parameter is not available (the data of the corresponding method are not loaded into the project), the type will be set to *VelocityP* automatically.

Values of the selected parameter are displayed in the form of labels on the layered model. At the same time, velocity values are indicated with colors according to the color scale, thus allowing to display values for two different methods simultaneously.

Par value – sets the initial value of the parameter in the first layer (for the current parameter type).

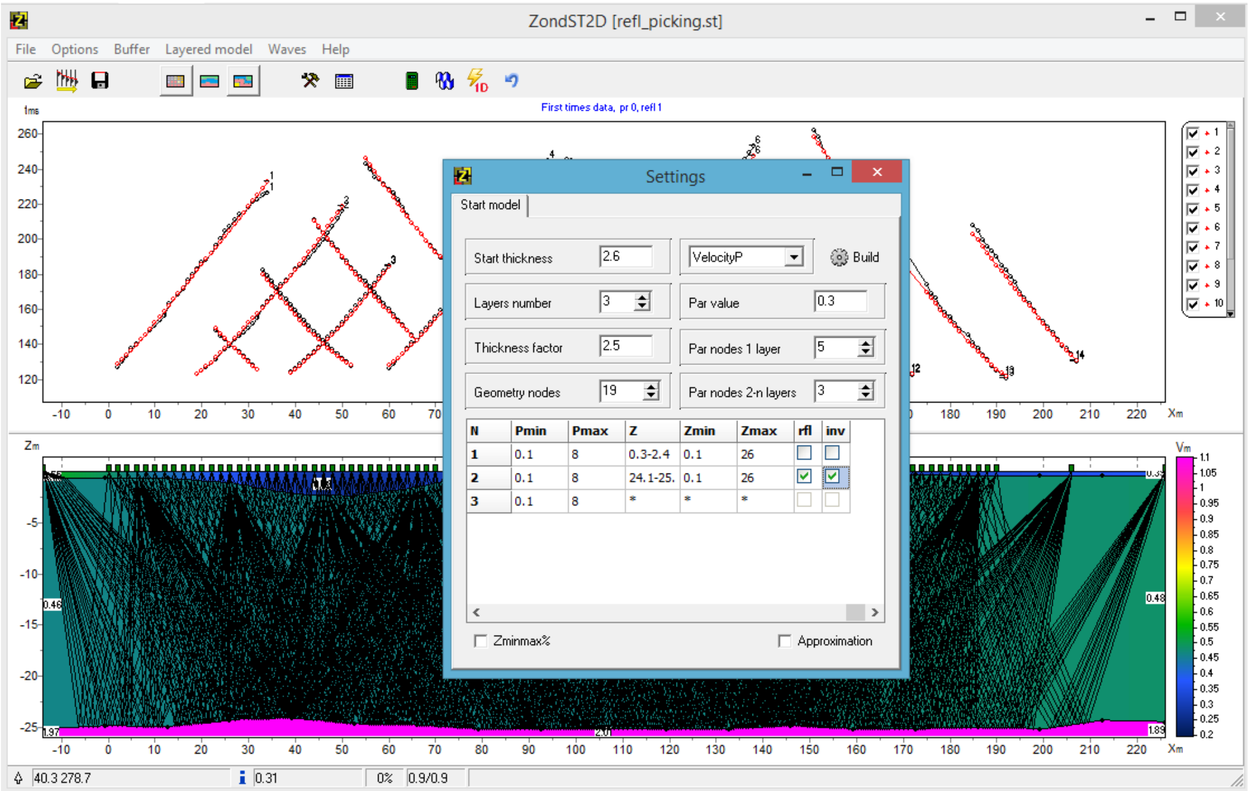
Par nodes 1 layer – sets the number of parameter nodes defining the parameter distribution within the first layer. The parameter variation profile is specified by a set of fixed node values with linear interpolation between the values. If set to 1, the parameter in the layer is set to a constant value. If set to 2, the parameter values are linearly interpolated from one edge of the model to the other.

Par nodes 2-n layers – sets the number of parameter nodes defining the parameter distribution within the subsequent layers.

The lower portion of the **Model constructor** window contains an editable table. In the table you can manually set the parameter value **par**, parameter variation range during inversion (from **Pmin** to **Pmax**), depth of the layer base **Z**, and depth variation range during inversion (from **Zmin** to **Zmax**). The depth variation range can be set as a percentage of the **Z** value (if the **ZminMax%** checkbox is checked), which simplifies the initial model creation, but the inversion will likely have to be run several times since the variation ranges set this way are typically too narrow.

Checking the **Approximation** checkbox allows calculation of approximate solution for the layered model, which implies that waves propagate strictly along refractive boundaries; this option increases the calculation rate.

If reflection data are present in the project, two additional columns – **rfl** and **inv** – appear in the table. To display these columns, one of the reflected wave modes should be activated (**Waves / Reflected – 1-3**). The program allows selecting one of the model boundaries as a reflecting boundary. For this, activate a checkbox in the **rfl** column; the base of the layer will be selected as reflecting boundary. To include the reflection data in the inversion, activate the corresponding checkbox in the **inv** column.



To perform a joint inversion of reflected and refracted wave data, the *Wave* type should be set to *Reflected* (**Waves / Reflected – 1-3**) and the **Layered / Invert VP&VS** option should be activated. For the *Reflected* wave type, the program allows displaying the reflected ray paths.

To perform a joint inversion of two types of data (other than seismic reflection), select the data type from the  drop-down list and configure initial models for each of the methods involved in the joint inversion.

After specifying the initial model parameters, press the **Build** button.

The **Layered / Invert boundaries** option allows the determination of layers geometry during inversion. To invert only parameter values (in the case when the boundaries are known and fixed), the **Invert boundaries** option should be disabled.

The **Layered / Draw labels** option shows/hides parameter values.

The **Layered / Transparent** option allows displaying the results of inversion in mesh mode in the background (the colors of the layered model become transparent).

The **Layered / Edit mode** enables the editing mode. The position of boundary nodes can be changed by dragging node points in the vertical direction, parameter values can be edited by right-clicking on the value labels.

A layered model can be saved to and loaded from a text file using the **Layered / Save layers** and **Load layers** menu commands.

The correctness of the initial model can be checked based on the ray coverage of the model. For this, solve the forward problem and look at the ray coverage (**Options / Data / Ray paths**) – the rays paths should pass through the boundaries of all layers.

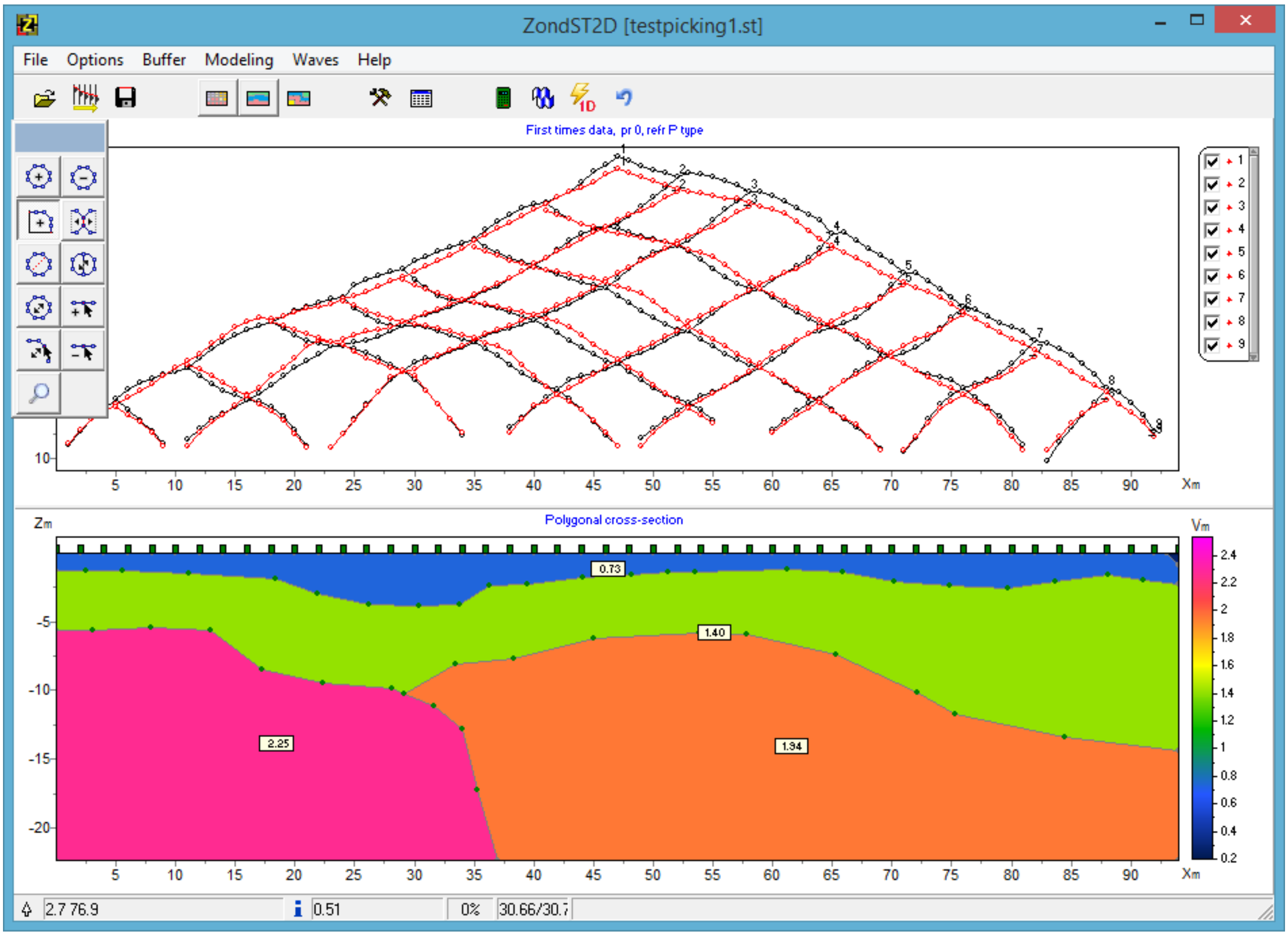
Initial model can also be determined automatically using one-dimensional inversion of travel time curves (the button on the toolbar). Right-click on the button brings up a context menu where you can select the type of inversion: *1D solution* – 1D solution is calculated for a travel time curve averaged along the survey line, *1.5D solution* – the solution is calculated for individual travel time curves along the survey line.

If any of the model parameters are known, they can be fixed for inversion in the **Layered / Edit mode** by left-clicking on the geometry nodes and right-clicking on the parameter value labels while holding down the Ctrl key. You can also exclude individual travel time curves from the inversion by deselecting them in the graphic plot legend. Deselected travel time curves will not be inverted if the **Options / Inversion / Invert visible data** option is enabled.

Polygonal model mode

In some situations, it is more practical to specify the model as a set of polygonal objects. Polygonal modeling contributes to more a structured model representation and simple management of elements.

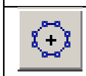
Clicking the button on the toolbar enables the polygonal modeling mode and brings up a floating toolbar with a set of modeling tools, which appears in the upper left corner of the screen.

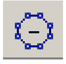
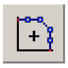
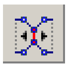
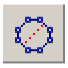
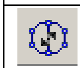
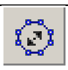
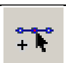
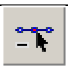
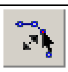


One way to create a polygonal model based on the results of inversion in the mesh mode is to plot the polygons over the mesh model displayed in the background. Before starting polygonal modeling, it is recommended to run the inversion and switch to the *Contour-section* visualization mode.

The exchange of parameter values between polygonal and mesh models is carried out using the **Modeling / Get values from mesh** and **Modeling / Set values to mesh** menu commands. The first option (embedding with the account of geometry) assigns the mesh model velocity values to already created polygons (using the results of preliminary inversion or modeling), and the second option assigns polygon velocities to the mesh model.

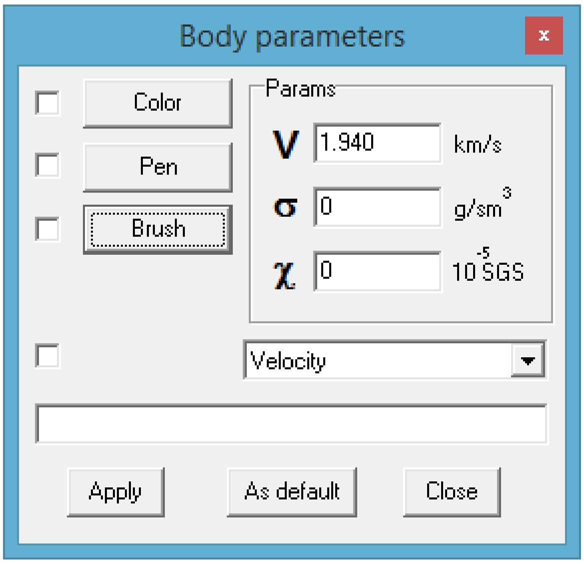
The mode allows the creation of both individual polygons (objects) in a homogeneous medium and a system of connected polygons (objects). Creating and editing of polygons is done using various tools from the floating toolbar:

Tool	Option
	Create a polygon. Left-clicking on the model adds a new node to the polygon. Right-click specifies the location of the last node and finalizes the creation of the

	polygon.
	Remove a polygon. Right-click on the polygon to remove it.
	Create a polygon coupled to an existing polygon or external boundary of the model. The first and the last nodes of the new polygon should be located either on the boundary of an existing polygon or on the external boundary of the model. The program will select the common boundary automatically or will prompt the user to select it if several options are possible.
	Disconnect coupled polygons to allow editing of individual polygons (moving, node editing, etc.). Left-click on the polygon to be separated (its boundary will change color). Right-click finalizes the uncoupling.
	Divide a polygon by a straight line (create two polygons from one). Click the left mouse button to indicate the first point of the line, then the right mouse button to indicate the second point. Both points should be on a boundary of the polygon to be divided.
	Move a polygon. Left-click on the polygon to capture it, right-click to release the polygon in a new location.
	Move a coupled polygon.
	Add a node. Right-click on the boundary to add a node.
	Remove a node. Right-click on the node you want to remove.
	Move a node. Left-click on the node to capture it, right-click to release the node in a new location.
	The Zoom mode to change the scale of the model section.

When working with the polygonal modeling tools, remember that **all actions are finalized by right-clicking**.

To modify polygon parameters, double-click on the polygon. The **Body parameters** dialog box will appear.



The V, σ, χ input boxes are used to specify the values of velocity, density and magnetic susceptibility.

The **Color**, **Pen**, **Brush** buttons bring up dialog boxes for setting the polygon fill color (different from the model's color scale), polygon boundary color and hatch pattern, respectively. If the corresponding checkbox is activated, the changes will be applied to all polygons in the model section.

The drop-down list contains the choices for polygon labels:

None – no label is displayed.

Velocity – velocity value is displayed.

All – velocity, density and magnetic susceptibility values are displayed.

User text – a user text that can be entered in the input box below is displayed.

Polygonal models can be saved using the **Save polygons** and **Load polygons** commands of the **Modeling** menu. With the **Export model to CAD** command, the polygonal model can be exported to the AutoCAD DXF file format.

Calculation of the forward problem for the created model is done by pressing the button on the toolbar or the Space key.

To increase the accuracy of the forward problem solution, it is recommended to set a more detailed model mesh. In particular, after creating a polygonal model, you can switch to the mesh mode and use the **Thin mesh** option in the properties of each of the axes, then return to the polygonal modeling mode and press the button. After this procedure, the calculation of the forward problem will be carried out using the more detailed model mesh.

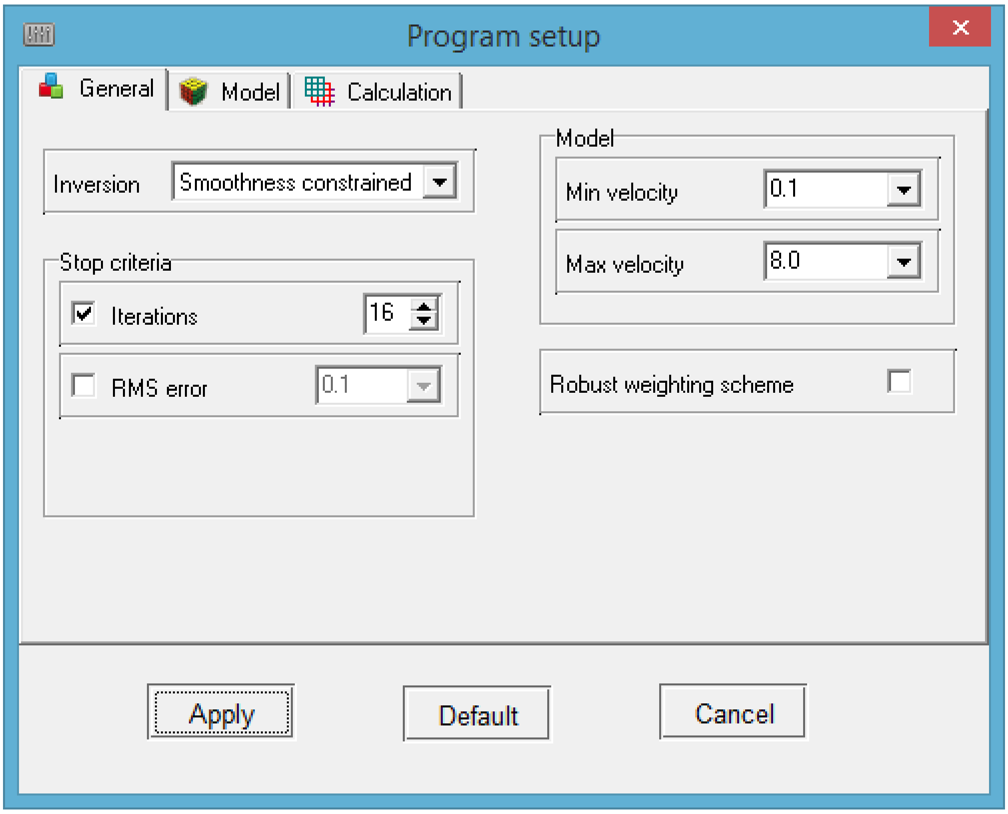
The **Polygonal modeling** mode can be used for joint interpretation of seismic data with gravity/mag survey data. In this case, the polygonal framework acts as a common part, i.e., the model geometry is fixed and only parameter values are inverted. Inversion of seismic velocities is started with the  button, and for the inversion of gravity and magnetic data, the **Options / GraviMagnetic / Inversion** command is used.

Data inversion

Changing inversion parameters

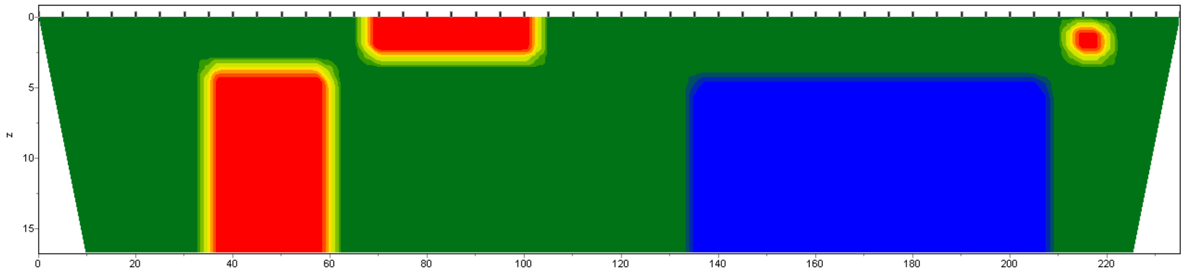
The dialog box for setting the parameters of forward and inverse modeling is opened using the  button on the toolbar or the **Option** / **Program setup** menu command.

The basic inversion parameters are configured in the **General** tab.



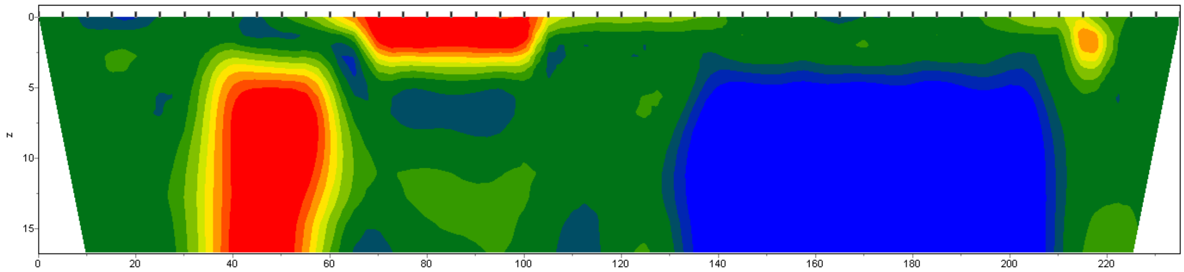
The **Inversion** option defines the inversion algorithm.

Let us consider various inversion algorithms on the example of a model consisting of several blocks (see figure below). The test model does not represent a typically observed distribution of velocities of elastic waves in the subsurface, but it allows us to clearly demonstrate the differences between the inversion algorithms.



To test the algorithms, we calculated the theoretical response for this model and added a five percent Gaussian noise to the result.

Smoothness constrained is a least-squares inversion using a smoothing operator. This algorithm results in a smooth (without sharp boundaries) and stable parameter distribution (see figure below).

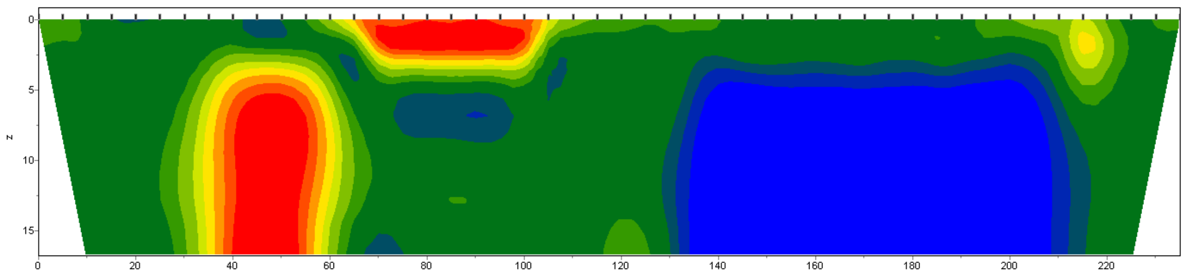


The matrix equation for this type of inversion is as follows:

$$(A^T W^T \mathrm{WA} + \mu C^T C)\Delta m = A^T W^T \Delta f$$

As follows from the equation, the program does not minimize the contrast of the model during the inversion. This algorithm allows achieving minimal misfits and is recommended for use at the initial stages of interpretation, in most cases.

Occam is a least-squares inversion using a smoothing operator and additional contrast minimization. This algorithm results in the smoothest parameter distribution (see figure below).

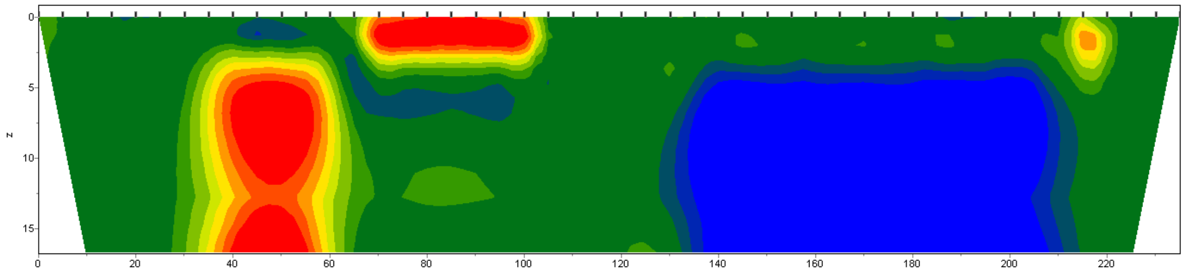


The matrix equation for this type of inversion is as follows:

$$(A^T W^T \mathrm{WA} + \mu C^T C)\Delta m = A^T W^T \Delta f - \mu C^T \mathrm{Cm}$$

The degree of smoothness of the resulting model is directly proportional to the **Smoothness factor** value. It should be noted that too large values of the parameter may lead to higher misfits.

Marquardt is a classical least-squares inversion algorithm with a damping parameter regularization (ridge regression). If the medium is characterized by a small number of parameters, this algorithm results in inversion models with sharper boundaries (see figure below).



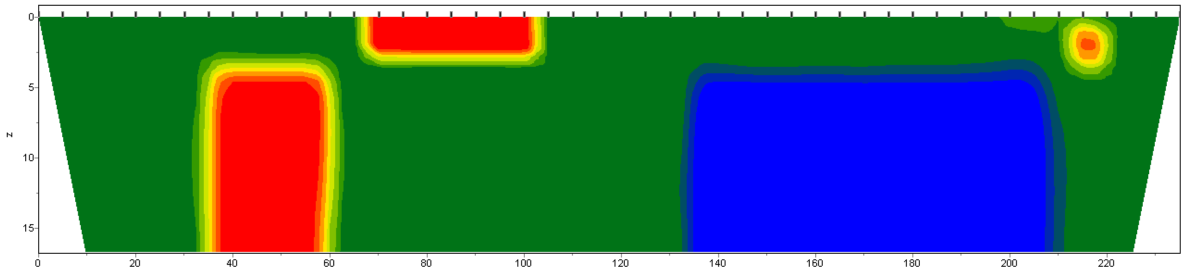
The matrix equation for this type of inversion is as follows:

$$(A^T W^T \mathrm{WA} + \mu I)\Delta m = A^T W^T \Delta f$$

Unconsidered use of this type of inversion can lead to unstable results or an increase in standard deviation.

It is recommended to apply the ***Marquardt*** algorithm as a refinement method (to reduce the misfit) after performing the inversion with the ***Smoothness constrained*** or ***Occam*** algorithms.

Focused is a least-squares inversion using a smoothing operator and additional contrast focusing. This algorithm results in a piecewise smooth parameter distribution, i.e., in a model consisting of blocks with constant velocity (see figure below).



The matrix equation for this type of inversion is as follows:

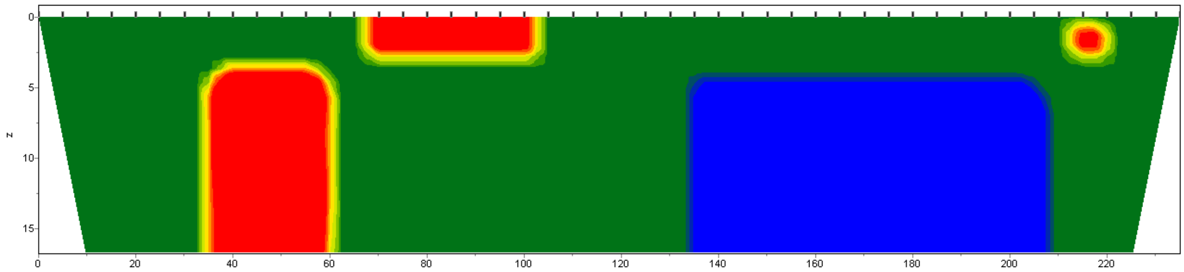
$$(A^T W^T \mathrm{WA} + \mu C^T \mathrm{RC})\Delta m = A^T W^T \Delta f - \mu C^T \mathrm{RCm}$$

Using this type of inversion requires a careful selection of the **Threshold** parameter in the **Model** tab. This parameter defines the contrast threshold value of neighboring cells, after reaching which the parameters of the cells are not averaged (i.e., it is considered that there is a boundary between the cells). The degree (weight) R_i of averaging of two neighboring cells is given by:

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

where e is contrast threshold and r_i is contrast between the cells.

Blocks – determination of parameters of individual areas that differ in velocity. Areas with the same resistivities are treated as single blocks (see figure below).



The matrix equation for this inversion type is the same as for the ***Marquardt*** algorithm:

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

It is recommended to use this algorithm at the stage of refining the model obtained with other algorithms (***Focused*** is preferred), having previously combined the cells into blocks using the **Cells summarization** module accessible by **Options / Extra / Model smooth/raster** menu command. The blocks can also be specified manually using the model editor to assign different parameters (velocities) to separate areas. Individual blocks are framed with a border while the **Program setup** dialog box is active.

The **Stop criteria** group box contains two criteria for inversion termination. The inversion process stops when one of the following criteria is met:

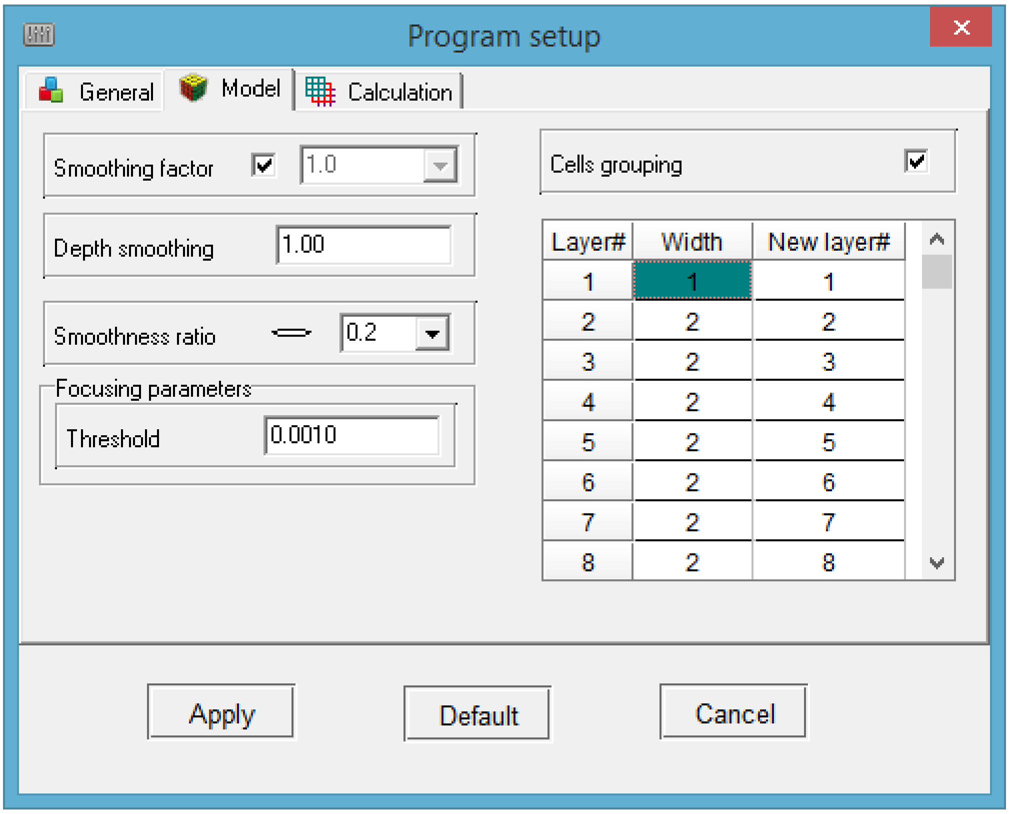
Iterations – the set number of iterations is reached.

RMS error – the set value of misfit is reached.

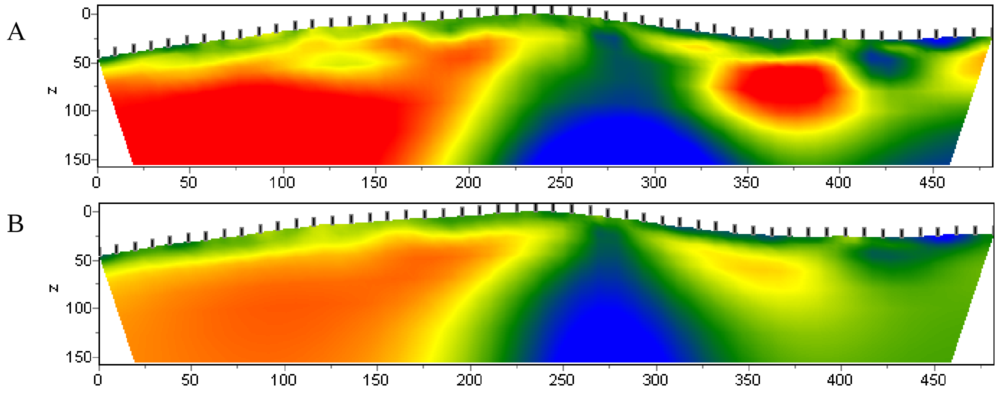
In the **Model** group box, the **Min velocity** and **Max velocity** boxes set the parameter variation limits. If overall parameter limits or individual cell limits are set too narrow, the inversion will repeatedly try to exceed the parameter limits. This can greatly affect the rate of convergence. In this case, the **Options / Inversion / Optimization / Lim based inv** option should be activated. Activating this option will reduce the contribution of cells exceeding the set limits, at the same time making such an exceeding difficult by applying a special parameter normalization.

Robust weighting scheme – this option should be enabled if there are spikes in the data associated with systematic picking errors. If the number of bad points in the data is comparable to the number of quality measurements, this algorithm may not produce good results.

The **Model** tab contains a set of inversion parameters related to models.

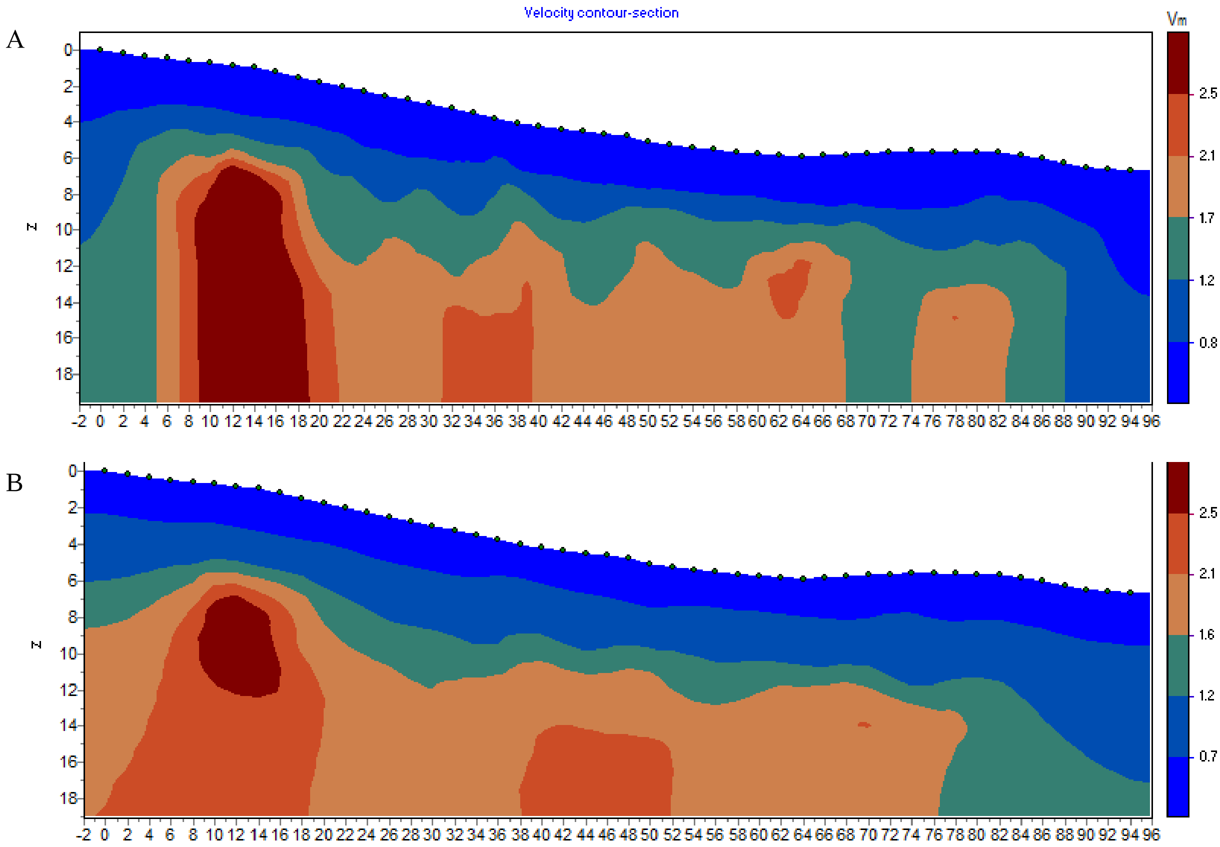


Smoothing factor – sets the ratio between the minimization of data misfit and model misfit. For noisy data or to obtain smoother and more stable parameter distribution, relatively large values of smoothing factor are chosen (0.5-10); for high-quality data, values in the order of 0.005-0.1 are typically used. Large values of the smoothing factor will typically result in larger misfits. The example models in the figure below were obtained using a smoothing factor of 0.01 for model A and 1.0 for model B (***Occam*** inversion). The resulting RMS errors were 4.5% and 6%, respectively. The smoothing factor is used in the ***Occam*** and ***Focused*** inversion algorithms. If optimization (Line search) is disabled, the program enables automatic determination of the smoothing factor. For this, the checkbox in the **Smoothing factor** box should be checked.

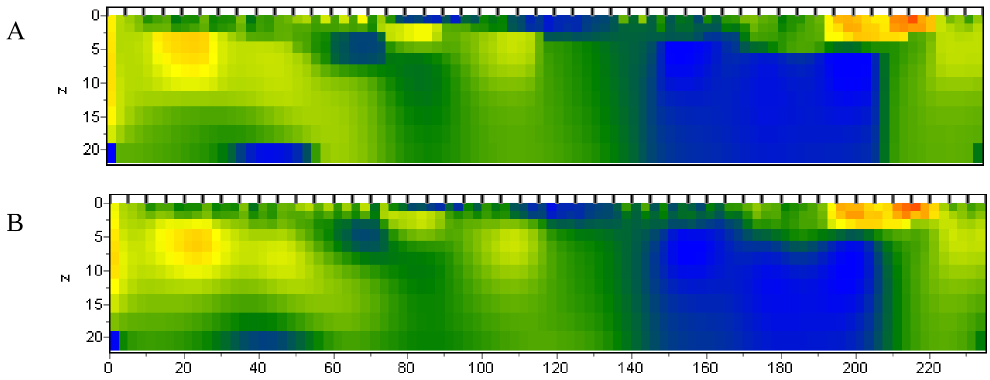


Depth smoothing – sets the coefficient of smoothing with depth. The value of this parameter will depend on the level of noise in the data; with high noise levels and appearance of oscillations and geologically unrealistic objects in the lower portion of the model, the smoothing value should be increased. The value is selected empirically.

Smoothness ratio – defines the ratio between the degree of smoothness in horizontal and vertical directions. Values of less than 1 should be used for horizontally layered geology and greater than 1 for vertically layered geology. Typically, the values range from 0.2 to 1 (in the figure below, model A was obtained with a smoothness ratio of 1, model B – with a smoothness ratio of 0.5).



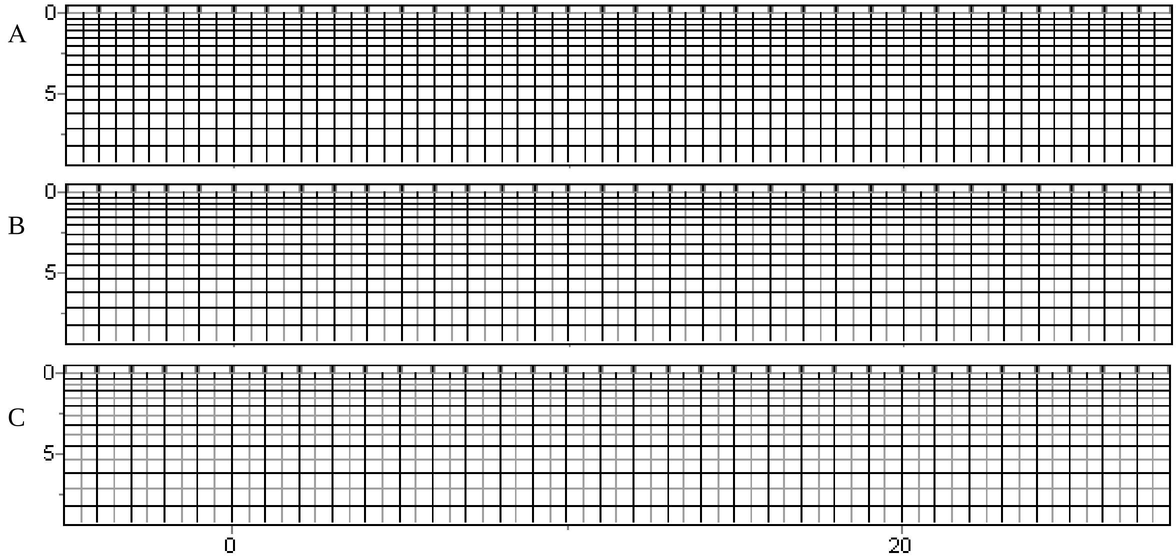
Threshold – sets the contrast threshold value of neighboring cells, after reaching which the parameters of the cells are not averaged (i.e., it is considered that there is a boundary between the cells). The value of this parameter is chosen empirically, typically in the 0.001 to 1 range. In the figure below, model A was obtained with a threshold value of 0.01, model B – with a threshold value of 0.1 (***Focused*** inversion). Choosing a very small value of this parameter can lead to an algorithm divergence (in this case the value should be increased). Large values of the parameter lead to a smooth distribution.



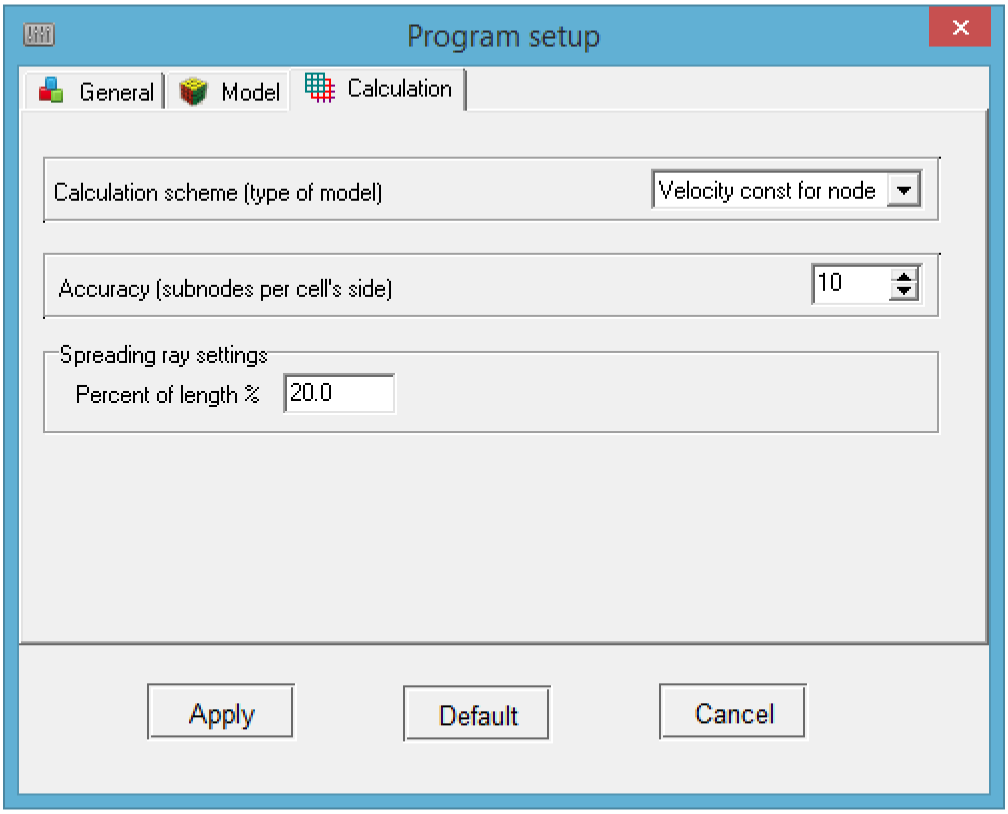
Cell grouping – this option is used in the case of large models. The option activates a table which allows you to combine adjacent cells (coarsen the grid), resulting in a smaller number of parameters defined during the inversion. While the number of cells used for inversion decreases, the number of cells used for solving the forward problem remains the same. Ideally, the number of parameters determined during inversion (thus, the number of cells) should be close to the number of data points.

The table contains three columns. The first column (**Layer#**) lists the numbers of the initial model layers. The second column (**Width**) specifies the width (in cells) of cells to be merged for inversion. The third column (**New layer#**) contains the numbers of the modified mesh layers and indicates the thickness of grouped cells. The modified mesh is displayed in the model section during its configuration. Double-clicking in the **Width** or **New layer#** columns allows merging cells horizontally or vertically. Right-clicking on a cell in the **Width** or **New layer#** columns brings up a dialog box where you can specify the number of horizontal cells or layers to be merged for the current and all subsequent cells in the column. Uncheck the **Cell grouping** checkbox to reset the table.

The figure below shows three examples of mesh configuration for inversion: **A** is the initial mesh configuration; for **B**, starting from the second layer, every two cells are merged in the horizontal direction; for **C**, in addition, starting from the second layer, every two cells are merged in the vertical direction.



The **Calculation** tab contains a set of parameters for forward modeling.

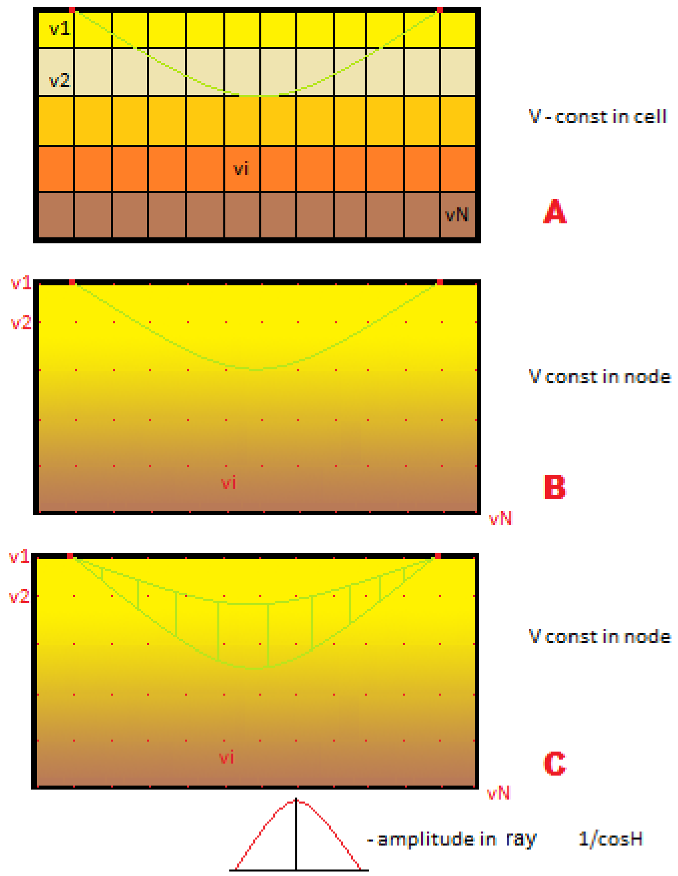


Calculation scheme (type of model) – defines the algorithm for solving the forward problem.

The program implements three variants of the forward problem solution, i.e. the calculation of seismic ray paths for an arbitrary 2D model (ray tracing). In the first variant (the *Velocity const* *for cell* option), the medium is divided into cells with constant velocities inside the cells; seismic waves propagate within a cell with constant velocity. This algorithm produces the least smooth ray coverage.

In the second variant (the *Velocity const for node* option), the medium is divided into cells with linear velocity changes inside the cells (the velocities are specified for cell nodes). This algorithm produces a smoother ray coverage compared to the previous one. Its drawback is in the transition from the node-specified velocity model back to cell (mesh) velocity model after inversion; this procedure smoothes the velocity values slightly and increases the misfit.

The third variant (the *Spreading ray* option) implements the most stable solution of the problem. It uses the concept of a ray spreading with depth, in which the radius of the beam expands as the wave propagates through the medium and captures more and more volume. Cells closer to the centerline of the beam have a stronger effect on the average velocity of the beam along the path. The **Percent of length %** option controls the degree of ray spreading as a percentage of the ray path length from source to receiver. The ray spreading algorithm is the most stable and produces the smoothest ray coverage, but it requires setting the spreading factor. In addition, it has the same drawback as the previous method related to velocity smoothing in the transition to mesh model.

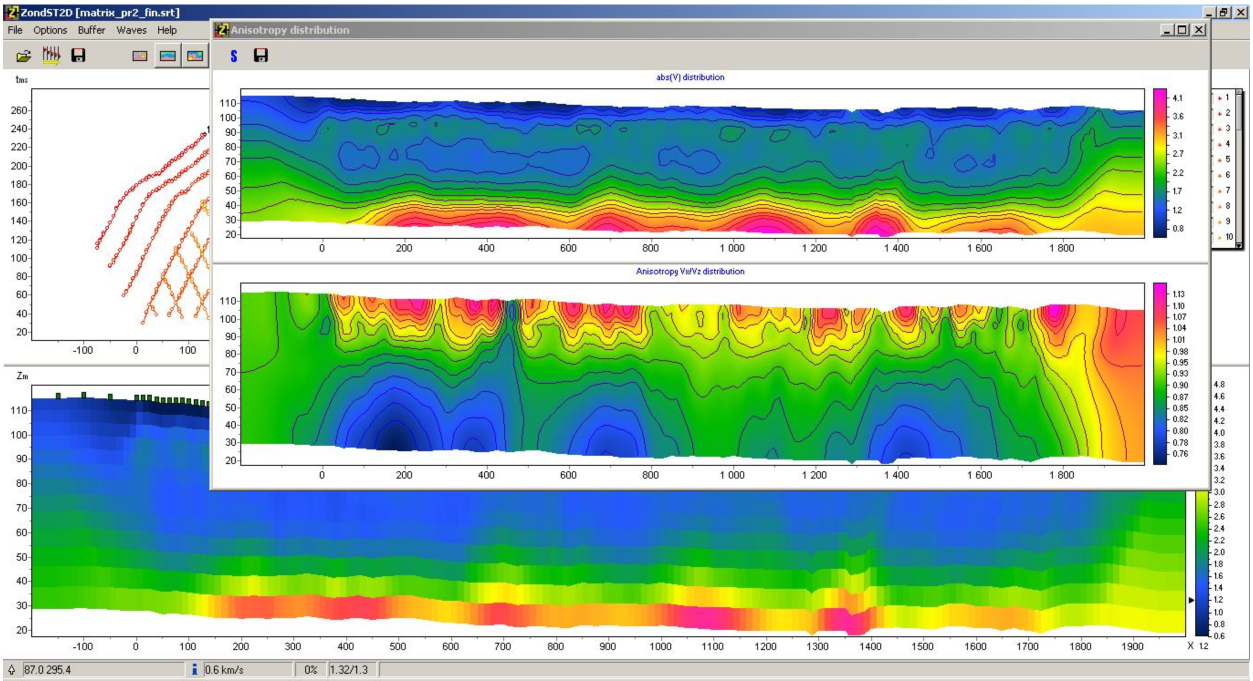


The **Accuracy (subnodes per cell's side)** option controls the calculation accuracy for all three algorithms. It sets the number of intermediate nodes used for the calculation, i.e., it determines the spatial accuracy. Typically, 10-20 intermediate nodes are sufficient for an accurate solution of the problem. Note that increasing the number of nodes decreases the calculation rate.

Inversion of velocity anisotropy

At the moment, a simple variant of accounting for seismic velocity anisotropy is implemented in the program. Different velocity values in a model cell in the vertical and horizontal directions are used for the calculations. Taking anisotropy into account has a particularly strong effect on the results of cross-hole measurements.

The program determines the average velocity and Vx/Vz ratio in the constant velocity mode in the cell. To calculate the anisotropy, select the **Options / Inversion / Invert anisotropy** option. When this option is selected, during the inversion (after the second iteration) a separate window will appear showing the distribution of the anisotropy parameter (Vx/Vz) in the cross-section and the calculated model. If it is necessary for the velocity section and the anisotropy section to have more similar features, it is possible to carry out a joint inversion in the cross-gradient mode. To do this, select the *Anisotropy* item in the *Cross-gradient* menu.



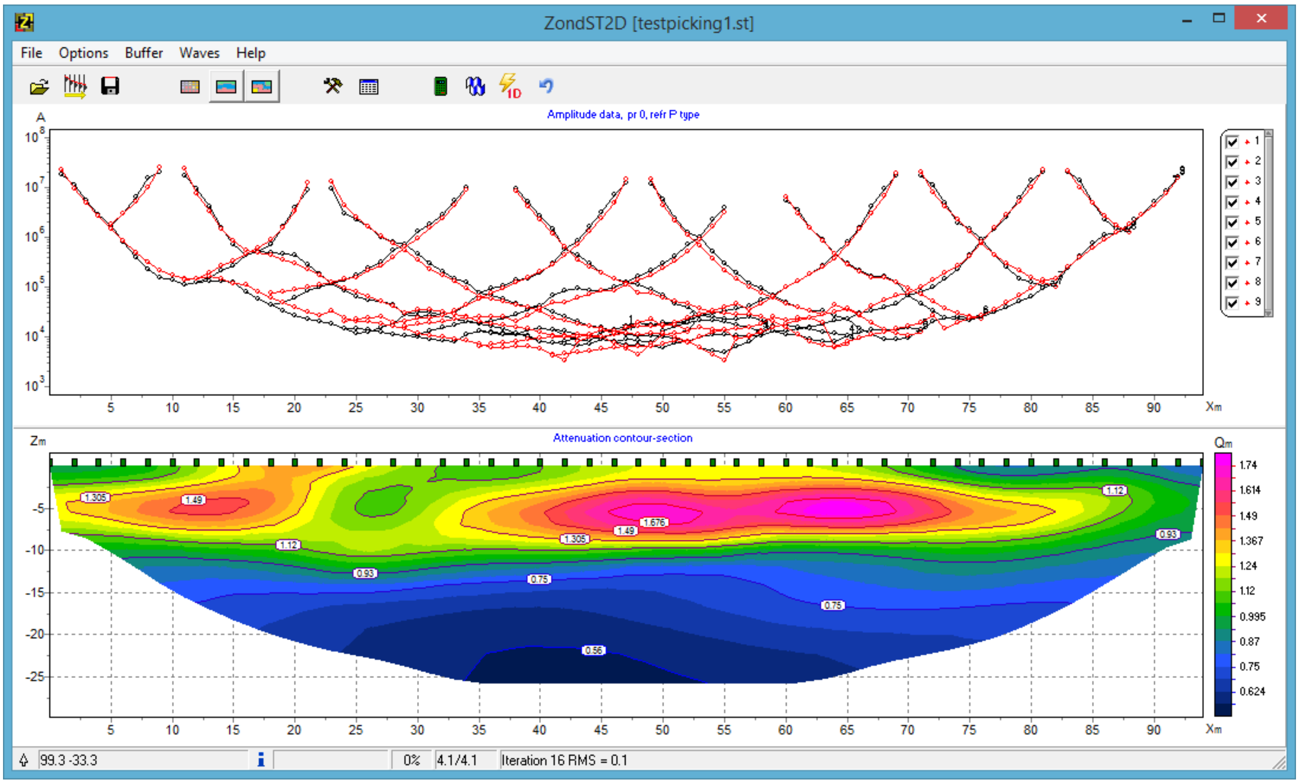
Care should be taken when performing the inversion of velocity and anisotropy simultaneously since it gives an additional degree of freedom (strengthens the equivalence) to the solution. It is recommended to perform such an inversion when you are certain that the velocity anisotropy is characteristic to the survey area or when interpreting cross-hole surveys, in which the "firing angles" can be quite wide.

To switch to the anisotropy modeling mode, select the **Options / Model / Display / Anisotropy** menu command. In this mode, you can model various geological conditions, assess the effect of anisotropy on the data, and estimate the possibility of anisotropy determination during inversion.

Attenuation tomography mode

The **Attenuation tomography** mode is designed to determine the attenuation (Q) of the medium (the parameter inversely proportional to Q-factor). To work with the data in the *Attenuation* mode, select the **Options / Model / Display / Attenuation** menu command. When the attenuation data are displayed, an additional *Attenuation* sub-section appears in the **Options** section of the menu.

The calculation algorithm assumes that the velocity inversion has already been performed and the ray coverage has been calculated.



The input data for the attenuation tomography are ray traces, 2D velocity model, and first arrival amplitudes. The amplitudes are automatically determined when picking the first arrivals. Note that the current algorithm realized in the program requires the maximum (absolute value) of the signal to be picked. If onset picking has been carried out, use the options for automatic amplitude collection in the *Collect settings* dialog box (**Settings / Amplitudes collect settings**) and **Options / Collect amplitudes from shot/all** commands of the **Trace editor** module.

Ray coverage is an important parameter for the inversion of amplitudes. Therefore, before running the inversion, make certain that the ray traces sufficiently cover the velocity model over the entire section.

In the *Attenuation* mode, the upper section of the main program window contains amplitude plots that are plotted similarly to travel time curves, with the logarithm of amplitude on the vertical axis. If the amplitude plots have a lot of peaks or look noisy, the data can be smoothed using the **Data / Smooth data** command. In general, the amplitude of a signal should decrease with increasing distance from the source.

The amplitude plots can be displayed individually or in groups using the legend and can be edited the same way as travel time curves (see the **Graphic plot** section for details).

The amplitude inversion algorithm uses the same basic settings as the velocity inversion. An important parameter that should be specified before running the inversion is the average frequency of the refracted waves. To set the average frequency (Hz), use the **Attenuation / Frequency** command. The frequency can be estimated visually or using the **Filter&Spectrum** module (see the **Data filtering** section for details). To start the inversion, press the button. During the iterative inversion process, the attenuation parameter (Q) model is displayed in the model section of the window, and the observed and calculated amplitude plots are displayed in the graphic plot section. The inversion result is the 2D distribution of the attenuation parameter.

A priori information

Inversion can be carried out taking into account a priori information. Incorporating a priori information into the inversion is realized, most often, in two ways: by setting the starting model (the presumed distribution and limits of its parameters) and by specifying the position of sharp boundaries.

By default, the starting model for inversion in **ZondST2D** is the current model displayed in the model section. It is updated by the program with each iteration during the inversion.

The known position of sharp boundaries available from borehole or geophysical data is most often used as a priori information. The boundaries are plotted in the boundary editing mode accessible through the **Options / Inversion / Set boundaries** menu command, which brings up a floating toolbar containing the following buttons:

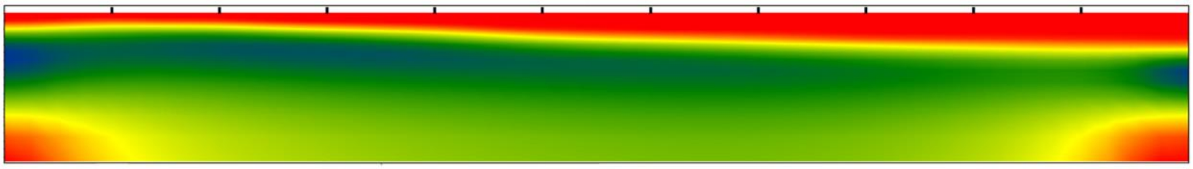
	Enable/Disable editing boundaries mode
	Add new boundary
	Delete boundary
	Save boundaries to file
	Load boundaries from file

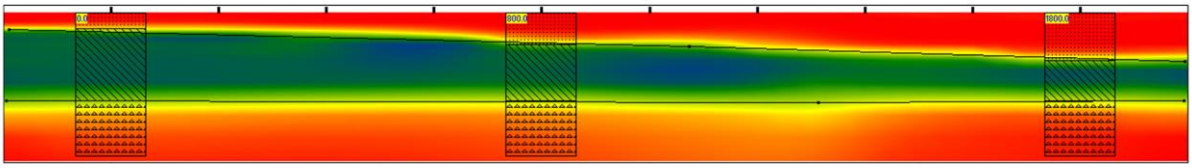
Introducing a priori geological boundaries into the inverse problem is an essential method to improve the quality of interpretation. It increases the stability of the problem while reducing the area of non-uniqueness and allows obtaining a more consistent structure. In the areas of the model where the parameters are only slightly sensitive to changes in the section, the incorporation of a priori boundaries is practically the only way to obtain acceptable results.

It is preferred to use the *Occam* inversion algorithm when a priori boundaries are incorporated into the model. Typically, only 1 or 2 boundaries are used; it is also worth noting that geological and geophysical boundaries do not always coincide.

It is recommended first to perform the *Occam* inversion and then plot the boundaries over the resulting model. When the boundary editing mode is enabled, boundary nodes are added by left-clicking on the model. Right-click adds the last node and finalizes the boundary. Avoid using too many nodes and plotting highly undulating boundaries; try to set the boundaries near the nodes of the model mesh. After plotting the boundaries, the inversion should be run again, this time taking into account the incorporated boundaries.

The top section in the figure below was obtained using the *Occam* algorithm without a priory information; the bottom section was obtained after performing the inversion with incorporated boundaries.



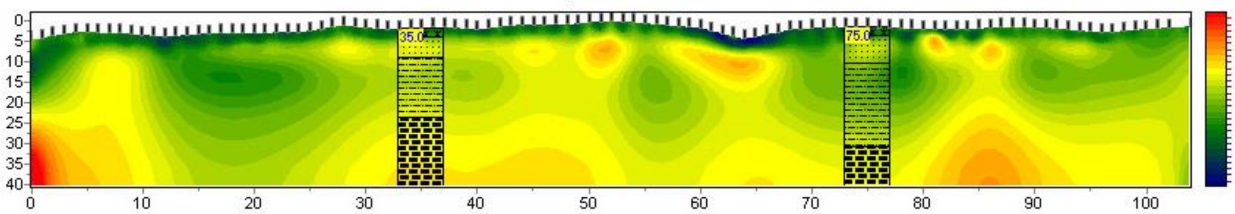


Another way of incorporating a priory information is realized in the **Image guided inversion** algorithm (**Options / Inversion / Cross-gradient / BG image**) which performs the inversion based on the similarity of the inversion model to a loaded graphic image.

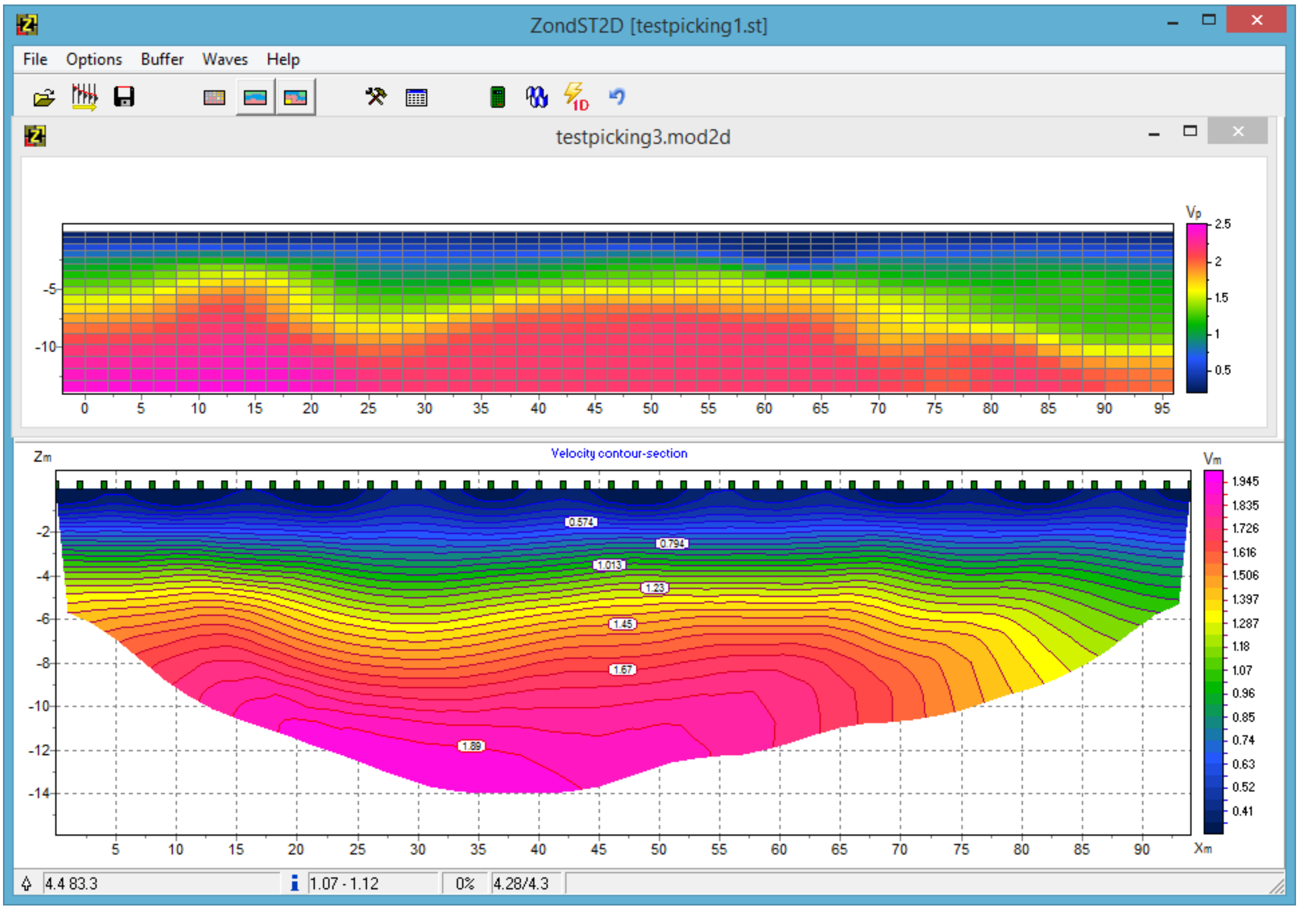
Several methods of visualizing a priori information are implemented in the program. Using the **Options /Import/Export** menu subsection, you can import a variety of geological and geophysical information:

- borehole columns;
- geophysical logs;
- horizontal profiling data in the form of graphs;
- models from other **Zond** programs;
- background graphic images (e.g., a geological or seismic section).

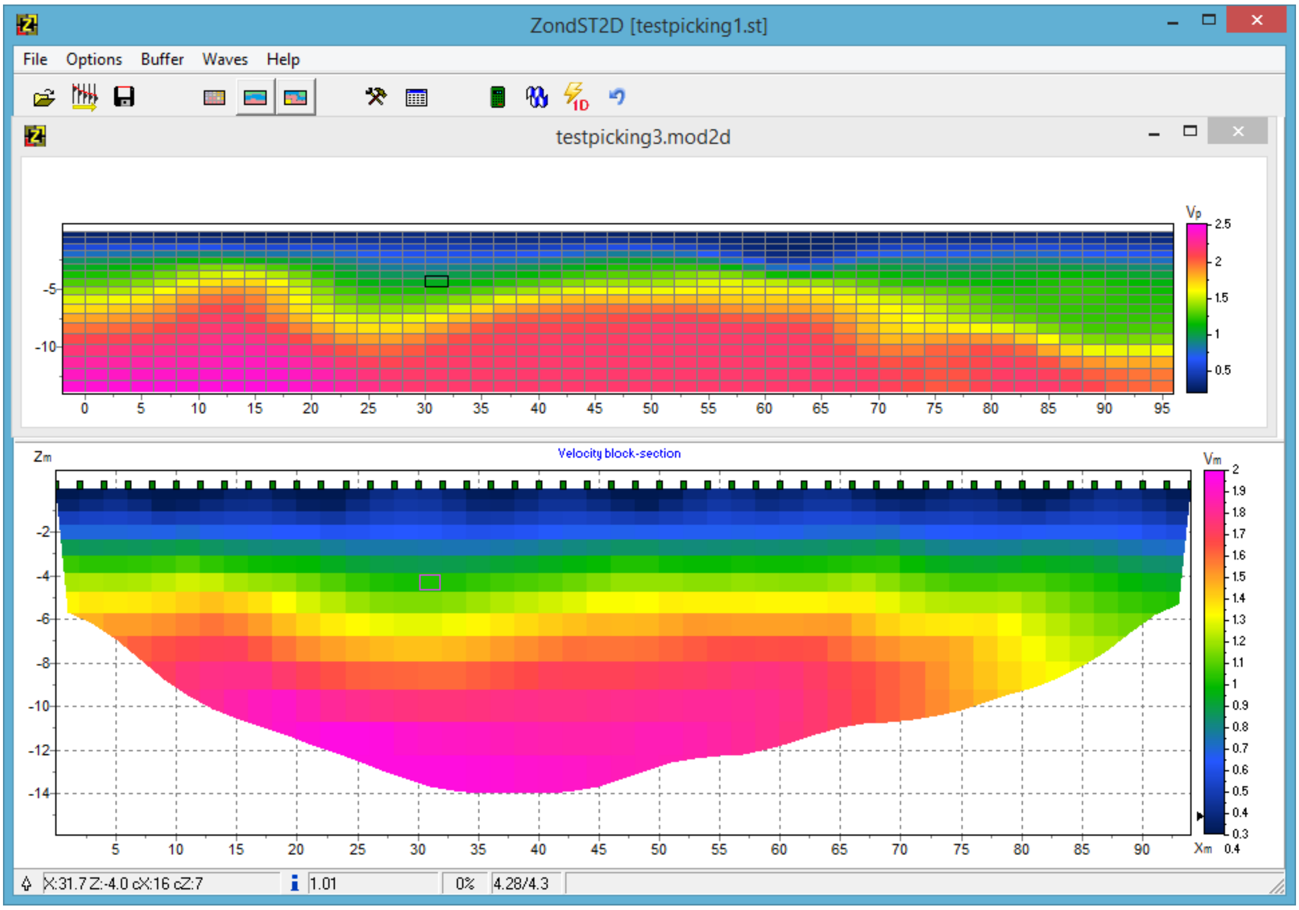
If borehole data are available, the columns and geophysical logs can be loaded into the model section using the **Options / Borehole / Load borehole data** command or created directly in the program interface using the editor.



The **Options / Import/Export / Import model/data** option allows you to open models created in **Zond** projects in separate windows. The option can be used to compare interpretation results of neighboring survey lines or when interpreting data from different methods.



When moving the cursor in the model area of the main program window, the corresponding cells of imported models opened in separate windows will also be highlighted, according to the size of the active cell.



Using the **Options** / **Import/Export / Import model/data** command, horizontal profiling data can be imported into the project and displayed in the form of a graph in the graphic plot area. The imported file should have the DAT extension and consist of two columns, with horizontal coordinates in the first column and measured values in the second. The profiling data will be plotted in the graphic plot area together with any seismic data opened in the project; an additional vertical axis will appear on the right.

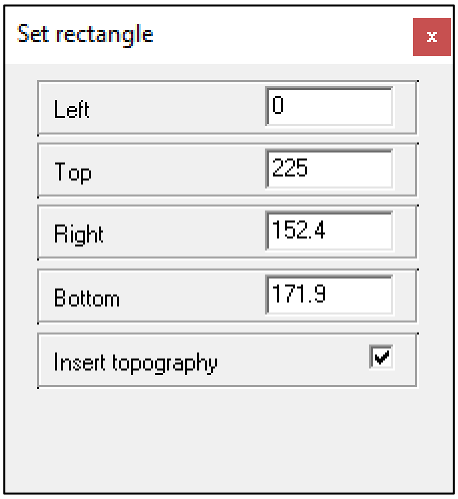
It is possible to save and load a fragment of the mesh model using the **Options / Import/Export / Model parts / Save selection** and **Load selection** commands. To save a fragment, select the desired fragment using appropriate selection tools and click **Save selection**. To load a model fragment, select a small area of the current model; the upper left corner of the selection will be a reference point for the fragment to be embedded. If there is no selection, the fragment will be inserted in the upper left corner of the model.

You can export and import a depth profile of the model parameter at a given horizontal location by using the **Options / Import/Export / Model parts / Extract 1d log** and **Load 1d log** commands. When exporting, the X coordinate of the depth profile should be specified in the export dialog box. When importing a depth profile, a range of X coordinates can be specified.

This option can be used, for example, for incorporating results of a downhole survey or to compare inversion results at the intersection of two survey lines.

A background image can be imported into the model section using the **Options / Import/Export / Background image** command. It could be a geological or geophysical section, e.g., a resistivity section or a velocity model obtained at a neighboring survey line. Graphic files in PNG and BMP formats and geophysical data files in the SEG-Y format are supported. Auxiliary spatial reference files in the internal SEC format can be used.

After selecting a graphic file (BMP or PNG), a dialog box appears in which you should specify coordinates of the image boundaries according to the coordinate system of the model section.



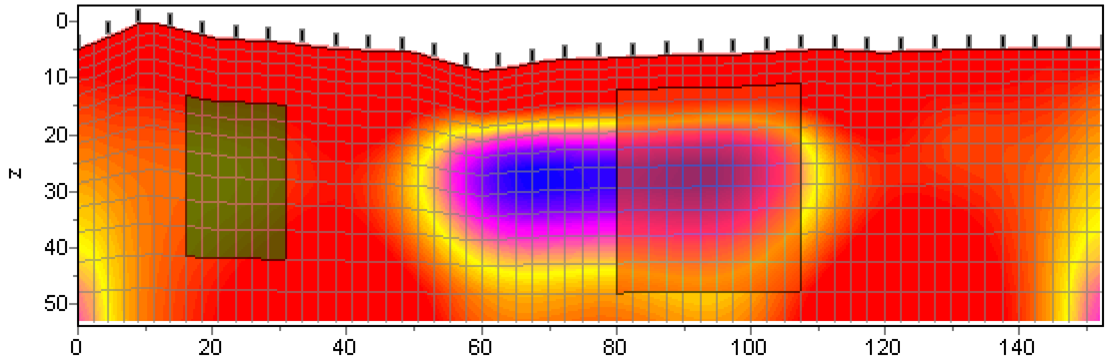
In this dialog box, you can manually set the coordinates of the top left and bottom right corners of the image.

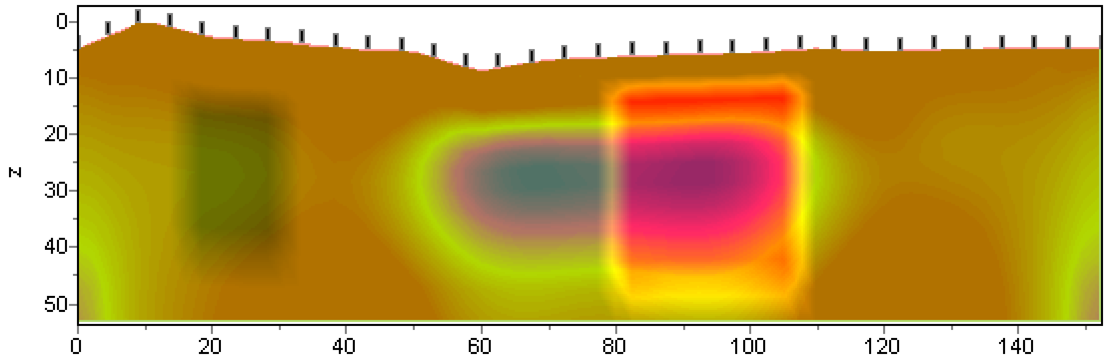
If the **Insert topography** checkbox is checked, the imported image will be transformed to reflect the topography of the model (the top boundary of the image will have the same geometry as the topography profile).

SEC is a reference file format, which is used to assign coordinates to imported images in accordance with the coordinate system of the model section. The file has the following structure: the first line contains the name of the image file, including the file format; the second line contains the four coordinates of the top left and bottom right corners of the image in the X1 Y1 X2 Y2 order, separated by space.

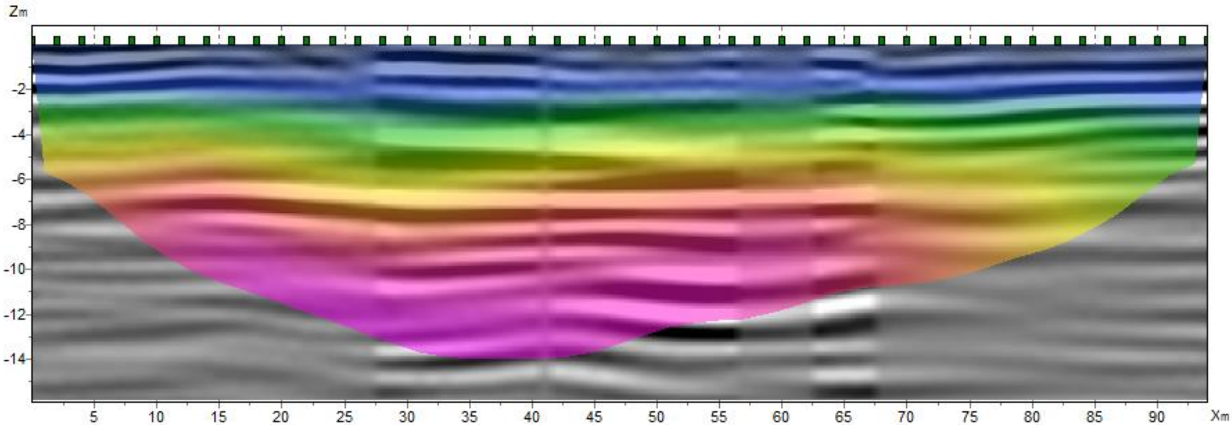
To change the transparency of the image, use the **Transparency** option in the **Model setup** dialog box (right-click on the title area of the model section or on the color scale, the **Setup** option in the context menu).

In the *Block-section* display mode, the cells with values differing from the host medium will stand out from the background image (the top section in the figure below). In this way, it is possible to model anomalous objects over the background. In the *Smooth-section* display mode, the colors of the background image and the current model will blend, giving you the opportunity to observe the features of the two sections at the same time (the bottom section in the figure below).



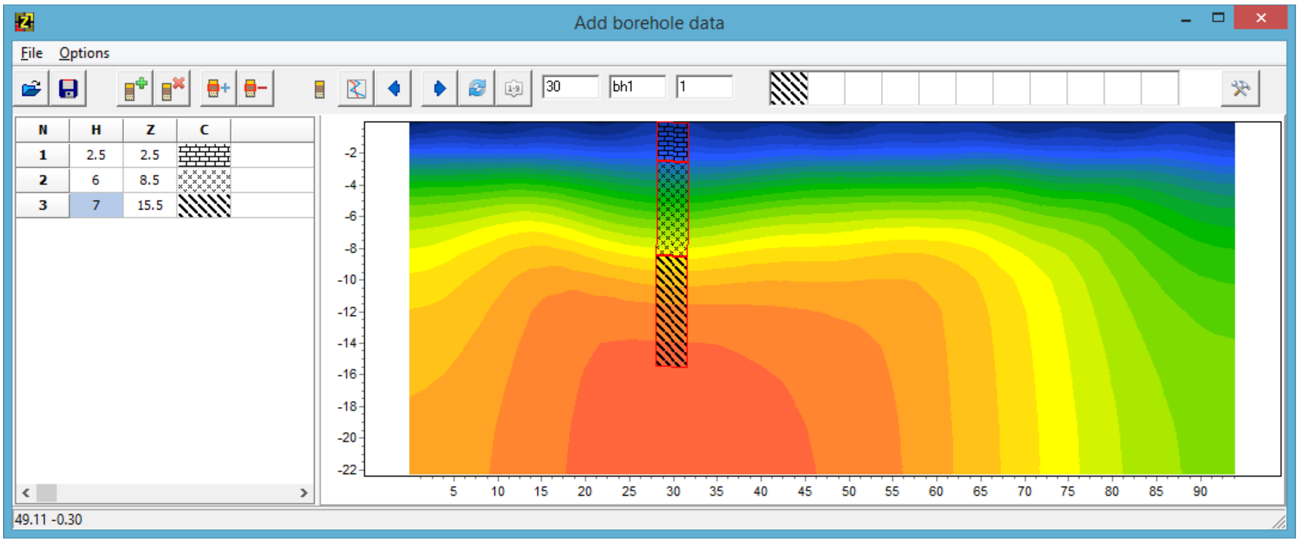


Importing a background image allows performing the data interpretation taking into consideration results of other surveys. For example, using sections obtained through seismic reflection surveys, ERT or geological mapping, a priori boundaries can be specified prior to data inversion. The figure below shows an example of a reflection image underlying a velocity model.

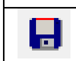
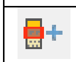


Creating borehole columns and logs

Creating borehole columns and geophysical logs is performed in a special module opened with the **Options** / **Borehole** / **Create/Edit borehole data** command. Using this module you can create, edit and visualize the borehole data along the survey line.



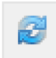
The module toolbar contains the following buttons:

	Open borehole file
	Save borehole file
	Create new borehole column or log
	Remove borehole column or log
	Add borehole column layer

	Remove borehole column layer
	Borehole column mode
	Logging data mode
	Go to previous borehole/log
	Go to next borehole/log
	Re-plot data plotting area
	Sort boreholes/logs by coordinate
30	Set horizontal coordinate
bh1	Borehole/log label (no more than 5 characters)
1	Borehole inclination angle
	Additional options

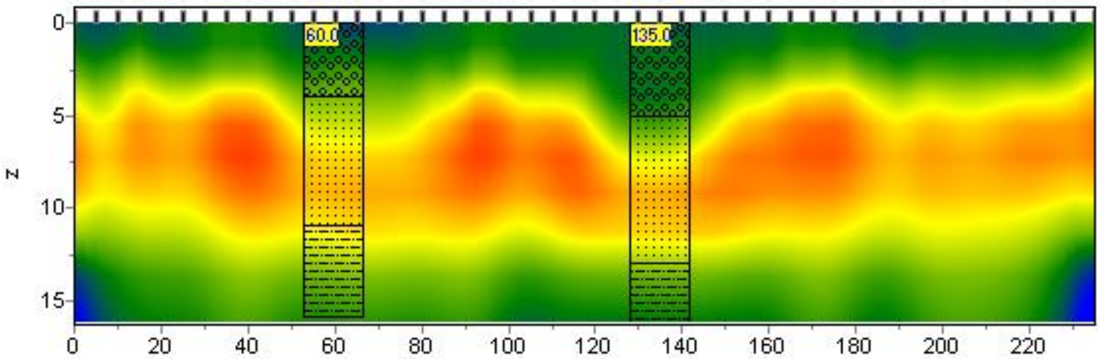
The module window is divided into two main sections. The left section contains a table with the following columns: **N** – layer number; **H** – layer thickness in meters; **Z** – depth to the bottom of the layer in meters; **C** – fill color / hatch pattern. The right section displays the graphical representation of the borehole data.

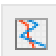
To start creating a borehole column, click the button on the toolbar. A new table will appear in the left section. Use the button to set the required number of layers, then specify the thickness or depth of each layer and choose the hatch pattern or fill color according to the borehole geology. Values in the **H** and **Z** columns can be edited by double-clicking on the desired cell. Double-click on a cell in the **C** column opens the **Pattern Color Editor** dialog box for selecting the hatch pattern. The program offers a vast selection of preset patterns; the **Color** button allows you to select a solid fill color.

After filling in the table, press the  button and the borehole will appear in the graphic section. The active borehole is displayed in red color. You can specify the borehole ID and horizontal coordinate using input boxes on the toolbar.

To facilitate working with a large number of boreholes, the program offers the possibility to create a palette. To create a palette, select the desired fill in the **C** column of the table, then right-click on a cell in the Fill area on the toolbar (a row of empty cells). You can create a set of fills, which can be saved and loaded using the **Save default palette** and **Load default palette** commands of the  context menu.

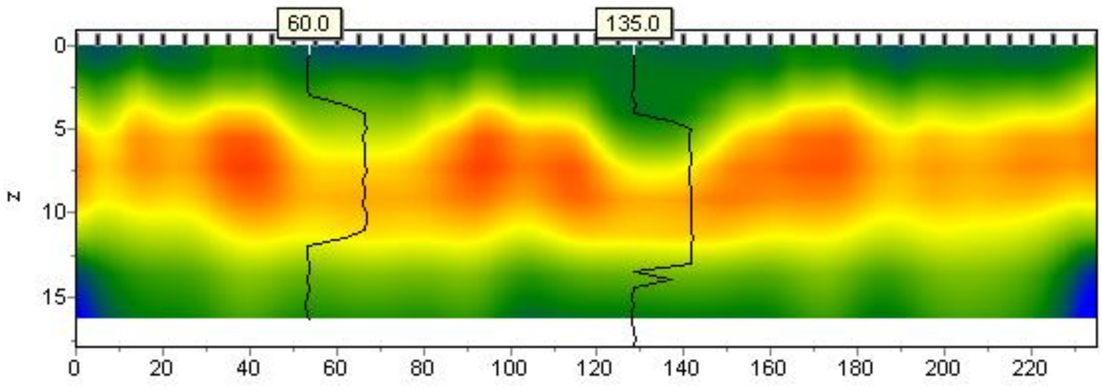
The Set borehole width command in the  context menu sets the borehole column width as a percentage of the survey line length.



Logging data are loaded using the  toolbar button (prior to this, a new borehole should be created using the  button). The supported file formats are TXT and LAS.

The text file should have the following structure: the first column contains depths from the ground surface, the second column contains measurement values, the third and fourth columns contain zeros. The columns are separated by Tab.

Below is an example of logging data displayed in the graphic section.



The created borehole columns and uploaded logs can be saved in the internal CRT file format using the button. The created CRT project file can then be opened in the main module of **ZondST2D** using the **Options / Borehole / Load borehole data** command.

Although the module provides convenient tools for the interactive creation of borehole columns, there is also a possibility to import borehole data in text format. For this, several text files should be created – one for each of the boreholes and one file that has a CRT format structure. When a project is saved in the module, the created CRT file is a binary file that cannot be opened in a text editor, however, it is possible to create a text file, change its extension to CRT and then open it in the module.

The following is a description of the CRT file structure for importing any number of borehole columns or logs into the module.

2280.txt // The first line is the name of the file containing a column or a log.

Skv2280 // The second line is the borehole label (will be displayed in the module).

18 2 2 1 0 1 0 0 // The third line contains the following parameters:

18 – X coordinate of the borehole;

2 – image width (as a percentage of the survey line length, typically 1-20);

2 – type of data (0-3);

0 – logging data (graph);

1 – logging data (interpolated color column) (the color scale of the section is used to display the data);

2 – borehole column;

3 – logging data (color column) (the color scale of the section is used to display the data);

1 – normalization parameter for logging data (0-2);

0, 1 – the same minimum and maximum values are used for all logs;

1, 2 – each log is normalized by its mean value;

0 – logging method index (if data of several methods needs to be displayed, enter different indexes for each of the methods starting with 0);

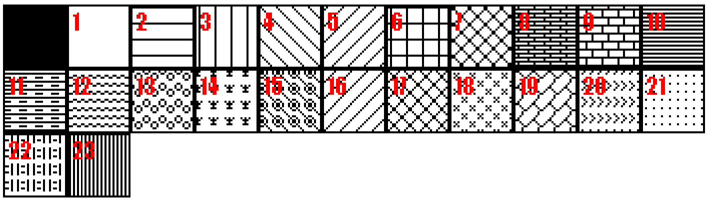
1 – graph color;

0 – scale (0 for logarithmic, 1 for linear);

0 – vertical displacement of the borehole relative to the ground surface.

The text file structure for logging data has been described above. For borehole columns, the following structure is used: the first column contains depths of layers (from the ground surface), the second column contains zeros, the third column contains values indicating fill colors for layers, the fourth column contains values indicating the type of hatch. The columns are separated by space.

Below is a list of fill colors and hatch patterns that can be used when describing a borehole column in a text file.



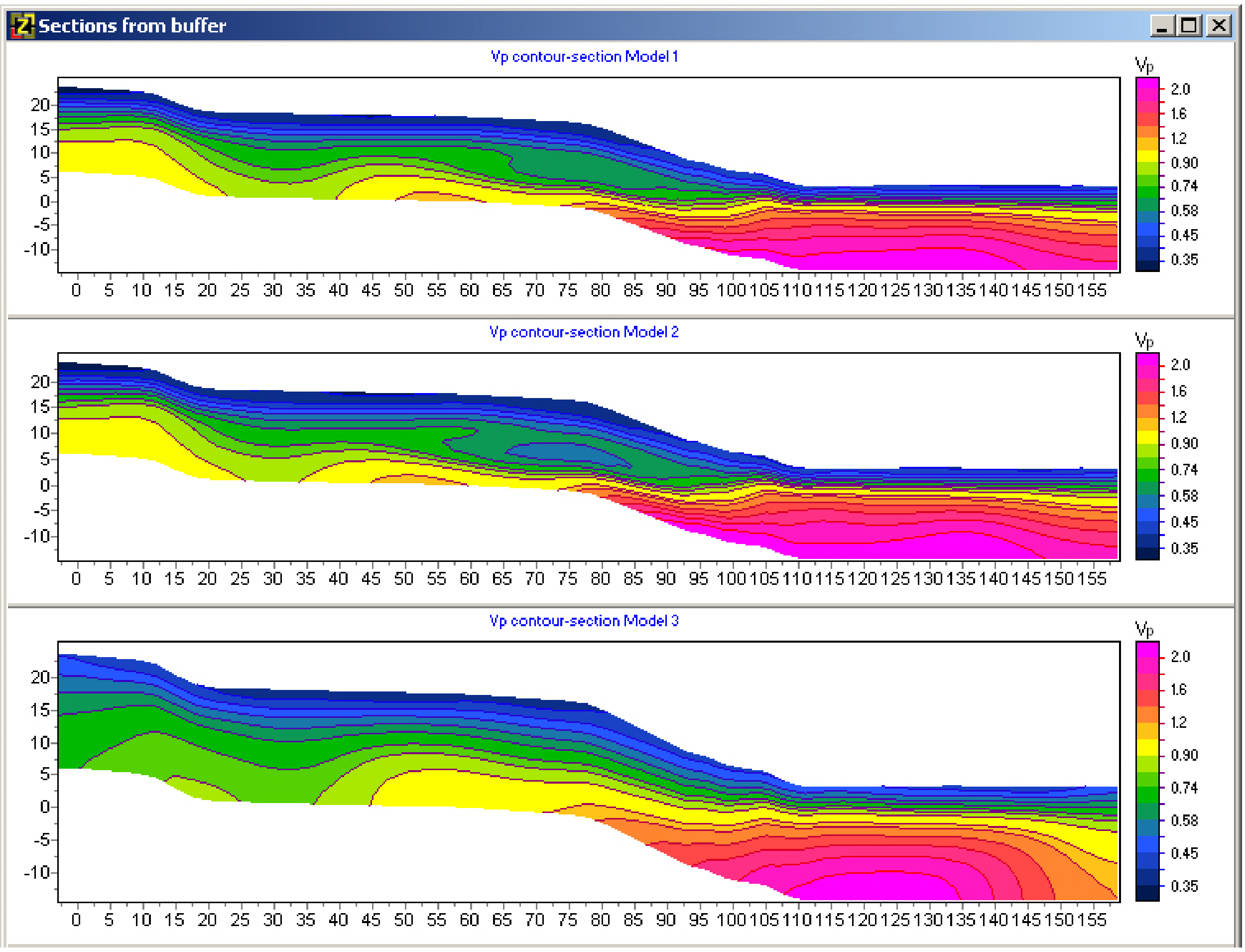
Interpretation results

Working with several models

It is often necessary to store several models within one project and display them simultaneously for comparison, e.g., when determining optimal inversion parameters, or when solving the forward problem for several similar models.

In **ZondST2D** current model can be saved into a buffer and displayed at any time using the **Buffer** section of the main menu. The **Model 1** through **Model 5** options correspond to the five buffer models that can be stored within one project.

To save the current model into the buffer, select one of the buffer models in the **Buffer** menu. If the selected buffer model is empty, the current model will be saved into this buffer slot. In the dialog box that appears you can enter the name of the buffer model which will be displayed in the **Buffer** menu and in the buffer models display window (see figure below) as a model title.



The buffer slots that contain stored models are indicated by checkmarks.

If the selected buffer slot is not empty (checked), in the dialog box that appears you can choose to load the buffer model (**From Buffer**) or to save a current model to this buffer slot (**To buffer**). If you choose **From Buffer**, the buffer model will replace the current model in the model section.

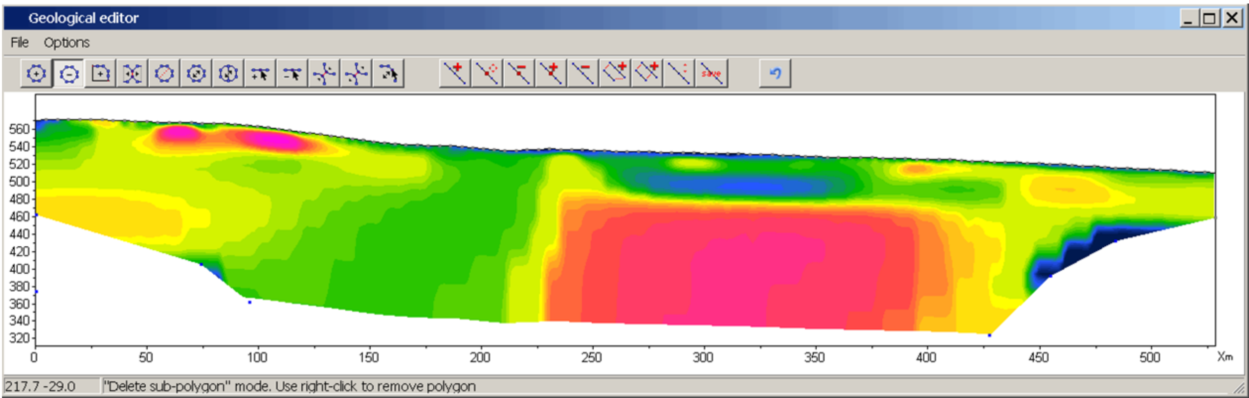
The **Buffer / Open** command opens a separate window where you can view and compare all buffer models.

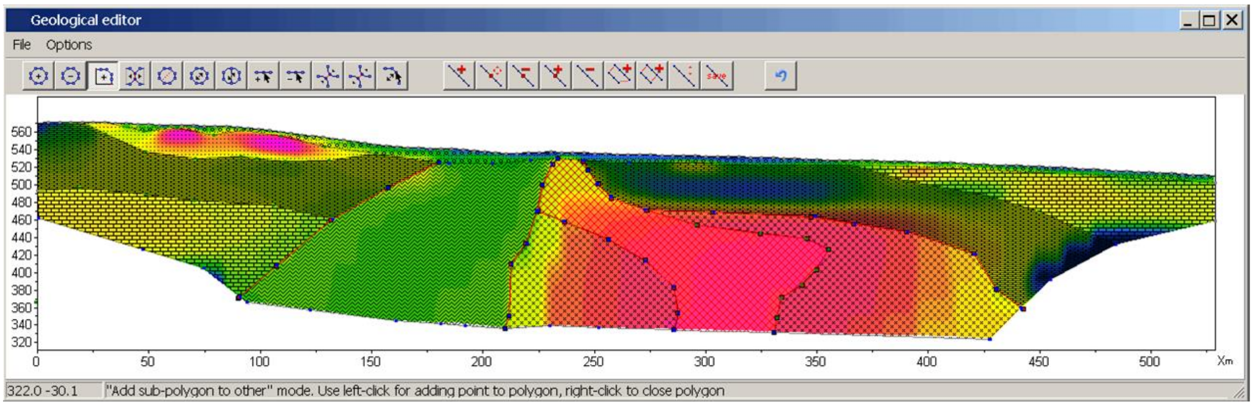
Geological editor

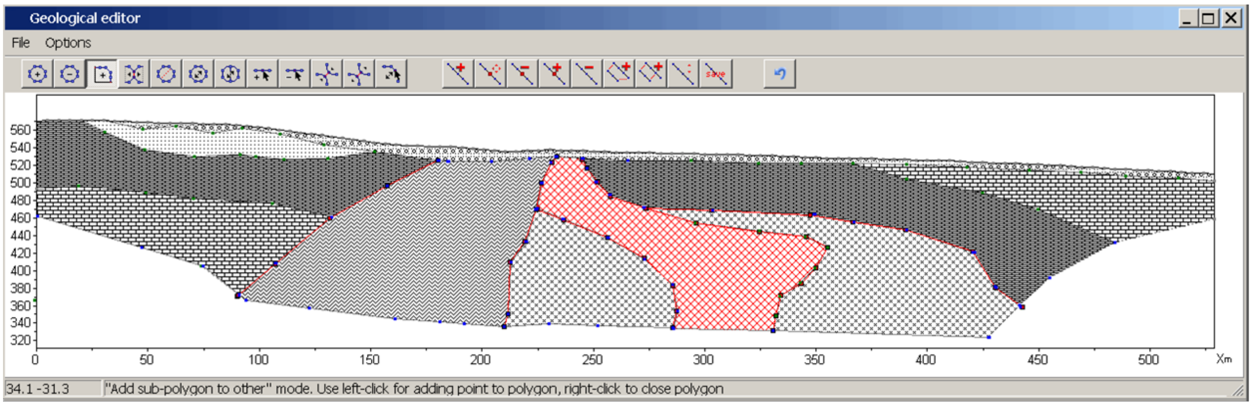
To build geological models (perform geological interpretation), a special **Geological editor** module (**Options / Geological editor**) is implemented in **ZondST2D**. The editor allows you to interactively create a geological model based on the current project model, borehole data, background images and models obtained in other Zond programs, as well as to print the resulting section at a user-defined scale, save and export the interpretation results.

In the editor the current inversion model is used as a background over which the geological model is plotted. When creating the geological model, local objects (polygons) and layers are plotted and then filled with patterns and/or colors corresponding to the geology. The module also allows displaying borehole data to simplify the model building process.

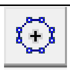
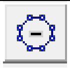
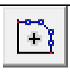
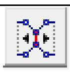
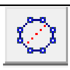
The main purpose of the module is a rapid creation of geological sections based on inversion results and their further export for reporting. Before running the module, the display mode and graphic settings of the inversion model have to be chosen. In most cases, contour representation of the model is the best choice.

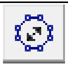
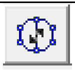
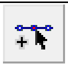
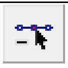
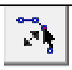
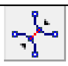
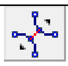






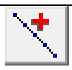
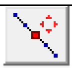
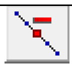
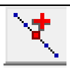
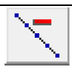
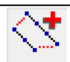
The toolbar of the **Geological editor** window contains a set of buttons for creating and editing polygons:

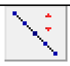
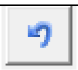
Tool	*Option*
	Create a polygon. Left-clicking on the model adds a new node to the polygon. Right-click specifies the location of the last node and finalizes the creation of the polygon.
	Remove a polygon. Right-click on the polygon to remove it.
	Create a polygon coupled to an existing polygon or external boundary of the model. The first and the last nodes of the new polygon should be located either on the boundary of an existing polygon or on the external boundary of the model. The program will select the common boundary automatically or will prompt the user to select it if several options are possible.
	Disconnect coupled polygons to allow editing of individual polygons (moving, node editing, etc.). Left-click on the polygon to be separated (its boundary will change color). Right-click finalizes the uncoupling.
	Divide a polygon by a straight line (create two polygons from one polygon). Click the left mouse button to indicate the first point of the line, then the right mouse button to indicate the second point. Both points should be on a boundary of the polygon to be divided.

	Move a polygon. Left-click on the polygon to capture it, right-click to release the polygon in a new location.
	Move a coupled polygon.
	Add a node. Right-click on the boundary to add a node.
	Remove a node. Right-click on the node you want to remove.
	Move a node. Left-click on the node to capture it, right-click to release the node in a new location. If the operation is not possible (there are intersecting boundaries), the node is returned to its original position. Nodes located on a model boundary can only be moved along the boundary.
	Couple two nodes belonging to different polygons. Left-click on the node to capture it, right-click to release it with the mouse pointer hovering over the node of the other polygon. The two polygons become coupled.
	Disconnect coupled nodes. Left-click on the common node of coupled polygons, right-click to release the uncoupled node in a new location.

The dialog box for changing graphic settings of a polygon is opened by double-clicking on the polygon.

The following toolbar buttons are used for creating and editing lines:

	Add a line.
	Move a node.
	Remove a node.
	Add a node.
	Remove a line.
	Create a polygon from two lines.

	Move a line.
	Save a line.
	Undo last action.

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

File / Load polygons – load polygons from a file.

File / Save polygons – save polygons of the current model into a file.

File / Show background – show the background image.

File / Remove background – hide the background image.

File / Print preview – open the **Zond Print Preview** dialog box.

Options / Model setup – open the dialog box for specifying the model area size.

Options / Load borehole data – load borehole data from a file.

Options / Remove borehole data – remove borehole data from the module.

Options / Remove all polygons – remove all polygons.

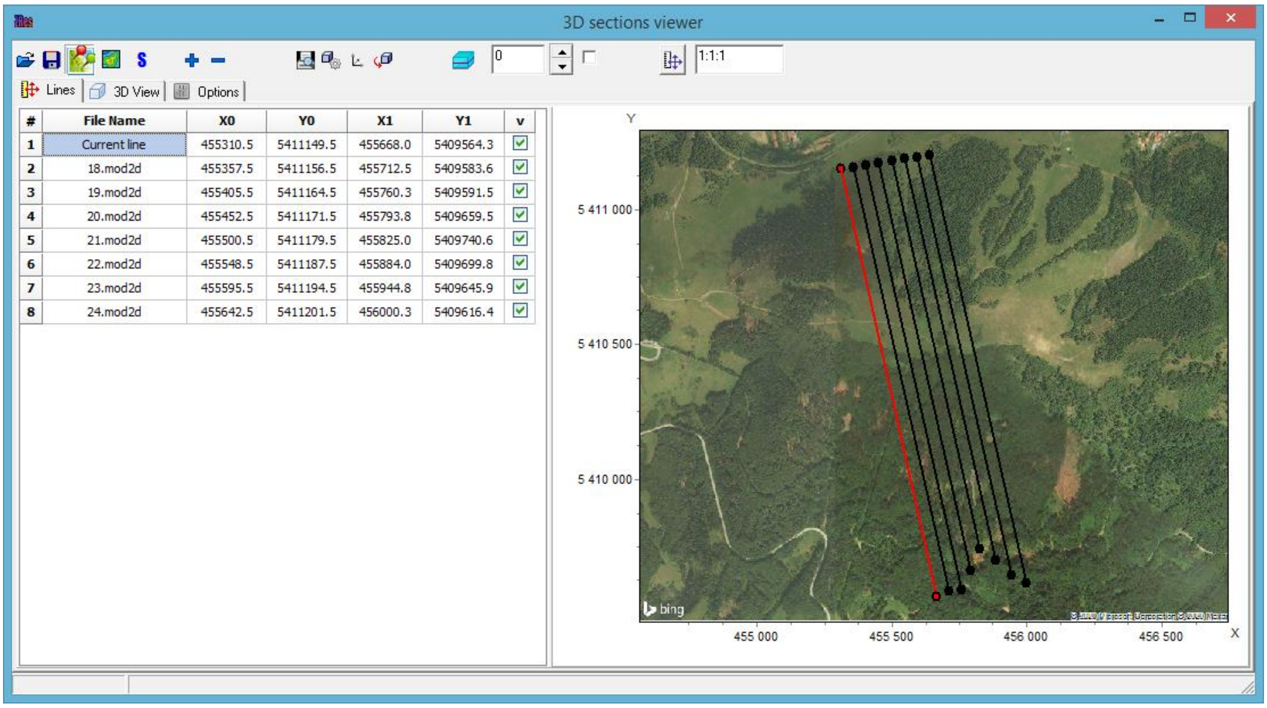
Options / Get from modeling – load polygons from the polygonal inversion model.

3D visualization of several sections

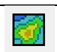
If the survey was conducted on a grid consisting of several closely spaced lines, a joint visualization and interpretation of these lines is recommended. This allows you to study the distribution of observed structures in the plan view and to facilitate the interpretation of each line by simplifying the identification of the most stable elements of the model.

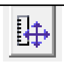
In **ZondST2D** the 3D visualization of velocity models is implemented in the **3D section viewer** module, opened with the **Options / 3D fence diagram** command. The module performs the interactive visualization of several models with their topography in a three-dimensional view and plotting of a parameter distribution at a user-defined depth/elevation in the plan view.

The **3D section viewer** window consists of three tabs – **Lines** for adding the 2D models and specifying survey grid coordinates, **3D View** for displaying the models in 3D and **Options** for image settings and scaling. The window toolbar provides access to various visualization options and commands for loading, saving and exporting 3D models.

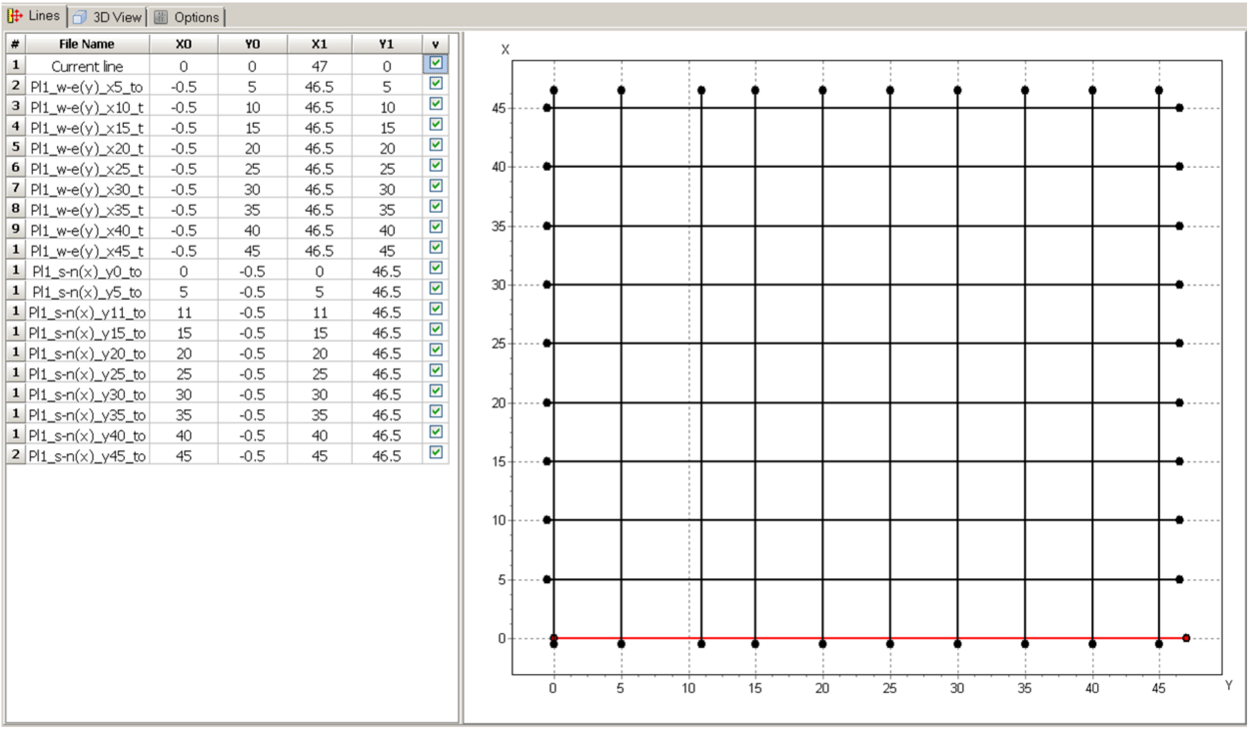


The toolbar contains the following buttons:

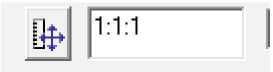
	Download a Bing satellite image from the Internet. The survey grid coordinates should be in the UTM coordinate system. If the image was not downloaded automatically, you should enter an actual Bing API key in the **Help / Bing maps api_key** dialog box.
	Build a horizontal (XY) slice at a specified depth.
	Build a horizontal (XY) slice in Surfer.
	Print preview.
	Open the 3D settings dialog box.
	Axes settings (see the **Axis editor** section).

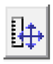
	Start 3D animation.
	Show a model slice in plan view. The depth of the slice is specified in kilometers in the input box to the right of the button: .
	Set proportional scaling for all axes (1:1:1). The input box to the right of the button allows you to specify the axes aspect ratio.
	Set the parameter for plotting (Vp or Vs)

The **Lines** tab is used for adding survey lines to the 3D plot and specifying their coordinates. Each survey line corresponds to a row in the table. To add a survey line to the table, right-click on a cell in the **File Name** column and select the model file to be loaded in the dialog box that appears. For the 3D visualization, the program uses files with the MOD2D extension which are created automatically when you save a project in the **ZondST2D** format. You can add an empty row to the table or delete a survey line using the  and buttons on the toolbar. The **X0**, **Y0**, **X1**, **Y1** columns of the table contain the rectangular coordinates for the start and end of the corresponding line (a curved line can be rendered if the loaded model contains XY coordinates in the topography profile). Any orientation of the survey lines is allowed – they can be parallel or non-parallel, intersecting or non-intersecting. The plan view plot of the survey grid is displayed in the right portion of the tab. The last column of the table (**v**) allows you to exclude lines from the 3D plot by unchecking the checkboxes.



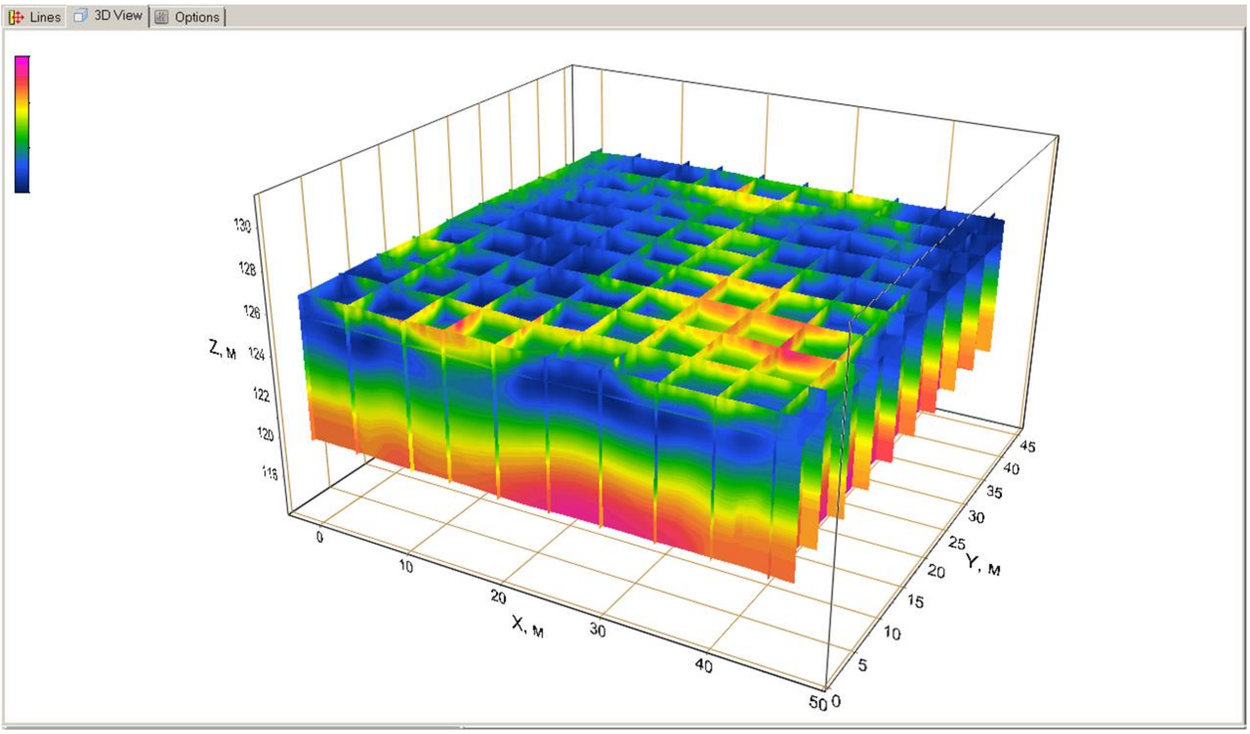
#	File Name	X0	Y0	X1	Y1	v
1	Current line	0	0	47	0	☑
2	PI1_w-e(y)_x5_to	-0.5	5	46.5	5	☑
3	PI1_w-e(y)_x10_t	-0.5	10	46.5	10	☑
4	PI1_w-e(y)_x15_t	-0.5	15	46.5	15	☑
5	PI1_w-e(y)_x20_t	-0.5	20	46.5	20	☑
6	PI1_w-e(y)_x25_t	-0.5	25	46.5	25	☑
7	PI1_w-e(y)_x30_t	-0.5	30	46.5	30	☑
8	PI1_w-e(y)_x35_t	-0.5	35	46.5	35	☑
9	PI1_w-e(y)_x40_t	-0.5	40	46.5	40	☑
1	PI1_w-e(y)_x45_t	-0.5	45	46.5	45	☑
1	PI1_s-n(x)_y0_to	0	-0.5	0	46.5	☑
1	PI1_s-n(x)_y5_to	5	-0.5	5	46.5	☑
1	PI1_s-n(x)_y11_to	11	-0.5	11	46.5	☑
1	PI1_s-n(x)_y15_to	15	-0.5	15	46.5	☑
1	PI1_s-n(x)_y20_to	20	-0.5	20	46.5	☑
1	PI1_s-n(x)_y25_to	25	-0.5	25	46.5	☑
1	PI1_s-n(x)_y30_to	30	-0.5	30	46.5	☑
1	PI1_s-n(x)_y35_to	35	-0.5	35	46.5	☑
1	PI1_s-n(x)_y40_to	40	-0.5	40	46.5	☑
2	PI1_s-n(x)_y45_to	45	-0.5	45	46.5	☑

The **Options** tab contains options for changing the display parameters – the color scale of the model and the scale for each of the axes. The **Continuous** option specifies whether the individual sections are plotted using contours or in the smooth (continuous) form. The axes aspect ratio can also be set using the 

input box on the toolbar. The  button allows switching between the proportional (1:1:1) and user-defined scaling.

The **Boreholes** option allows displaying borehole data in the 3D image. If the projects contains a substantial number of boreholes, rendering them might take considerable time.

The 3D model is displayed in the **3D View** tab.

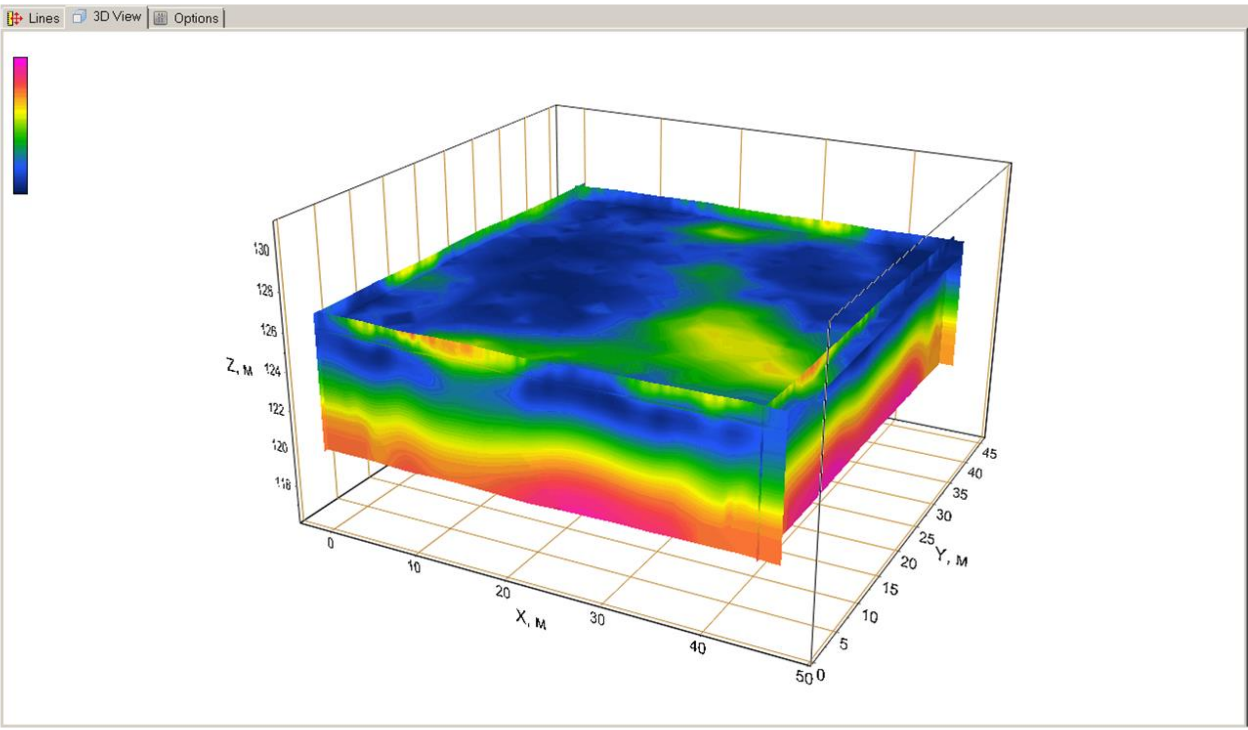


For each axis, the axis properties can be changed in the dialog box opened using the toolbar button. You can configure the axis labels and grid lines, change the axis title, etc.

The button gives access to the **3D View** display settings (projection types, scale, rotation angle, the position of the image on the screen, etc.).

You can rotate the 3D plot by pressing and holding down the left mouse button; the mouse wheel can be used to zoom in and out. Press the toolbar button to start the automatic rotation of the 3D plot around its geometric center.

The 50 box on the toolbar allows you to build and display a horizontal slice. The checkbox on the right defines whether the depth from the ground surface (when checked) or the elevation (when unchecked) of the slice is specified in the input box.



The button builds a horizontal slice for the specified depth/elevation.

The button builds a horizontal slice for the specified depth/elevation and exports the results to Surfer.

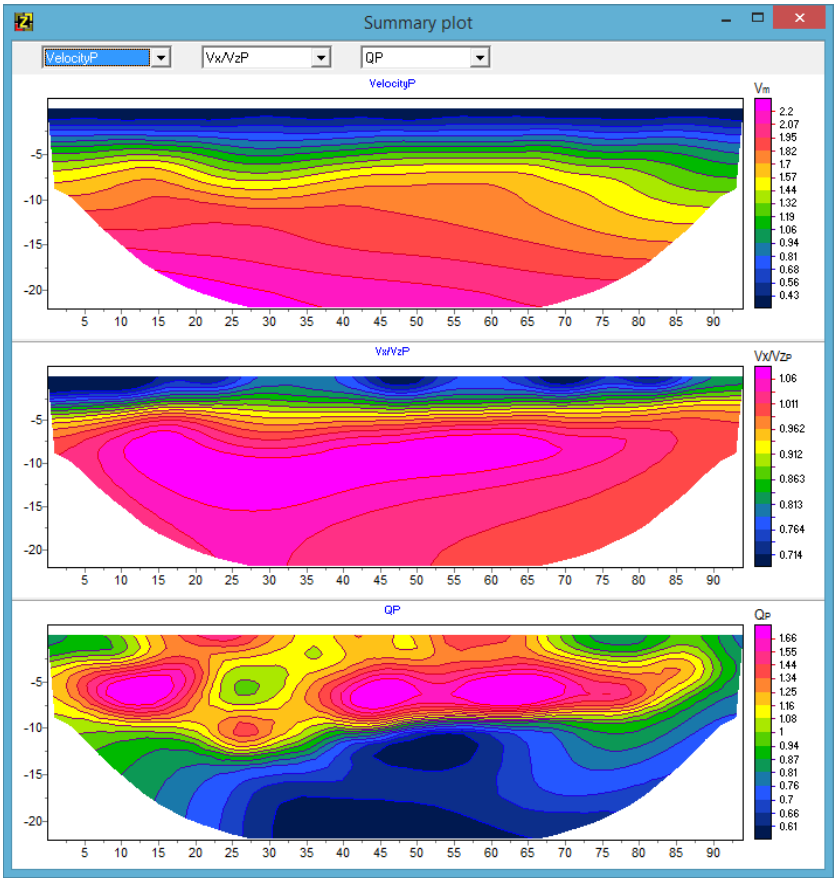
The button allows you to download a Bing satellite image from the Internet and use it as a background. The survey grid coordinates should be in the UTM coordinate system.

The current set of models in the **3D section viewer** can be saved and opened again later using the and buttons. If you select the **Project file** type in the **Save As** dialog box, a PRJ file is created, which contains full paths to individual MOD2D models and their coordinates. If you select the **XY plane** type in the **Save As** dialog box, the program will create a DAT text file for the current horizontal slice which can be used in other data mapping programs such as Surfer. When saving in the **Voxler 3d grid** format, the program creates a DAT text file that contains the entire model data. The 3D model workspace can be printed by pressing the toolbar button.

Summary plot

The summary plot is intended for joint visualization of various seismic parameters in a separate window. It is opened with the **Waves / Summary plot** menu command. Depending on which data sets are available in the project, the three drop-down lists in the upper panel of the window will contain various seismic parameters that can be plotted in the corresponding sections.

In the figure below, the summary plot window contains cross sections of P-wave velocity, velocity anisotropy and attenuation.



The full list of parameters that can be plotted is as follows:

VelocityP – P-wave velocity mesh model;

VelocityS – S-wave velocity mesh model;

Vx/VzP – anisotropy coefficient of P-wave velocities;

Vx/VzS – anisotropy coefficient of S-wave velocities;

QP – attenuation coefficient of compressional wave;

QS – attenuation coefficient of shear wave;

VP layered – P-wave velocity layered model;

VS layered – S-wave velocity layered model;

VS MASW – S-wave velocity from MASW data;

Poisson ratio MASW – Poisson's ratio from MASW data;

E MASW – Young's modulus from MASW data, $E = \rho \cdot \mathrm{Vs}^2 \cdot \frac{3 \cdot \mathrm{Vp}^2 - 4 \cdot \mathrm{Vs}^2}{\mathrm{Vp}^2 - \mathrm{Vs}^2}$;

G0 MASW – shear modulus from MASW data, $G0 = \rho \cdot \mathrm{Vs}^2$.

Saving the results

The project data are stored in a binary file of the internal **ZondSTD** format with the ST extension. The file stores the field data, measurement weights, the current model and much more. When the file is opened in the program, the current model is restored from the file's data.

The project data can be saved by pressing the button on the toolbar or selecting the **File / Save file** menu command. In the dialog box that appears you can choose to save the whole project, its parts in the form of observed or calculated data values, or an image of the entire workspace or model only in the BMP format. The model image scale and resolution can be changed in the **Picture settings** dialog box opened with the **Options / Graphics / Bitmap output settings** command. In the dialog box, you can specify the **Vertical scale**, **Horizontal scale** (in meters per centimeter), **Print resolution** (in DPI) and **Font size** for the export image. To apply the settings, disable the **Automatic** option, otherwise, the exported image will have the same dimensions as it appears on the screen.

The table below contains the description of file formats for saving and exporting the data:

Zond project data [*.st]	Save the project to a project file.
Zond calculated data [*.st]	Save calculated data to a text file.
Zond observed data [*.st]	Save observed data to a text file.
Zond model with calculated [*.st]	Save calculated data values, current model and all program settings to a project file. In this mode, the observed values are replaced by calculated values. This might be convenient for testing the inversion routine on different models.
Worksheet [*.bmp]	Save the workspace of the program window in the BMP image format.
Model [*.bmp]	Save the model section in the BMP image format. To change the scale and resolution, use the **Picture settings** dialog box (**Options / Graphics / Bitmap output settings**).
Program configuration [*.cfg]	Save the program inversion parameters.
Grid file [*.dat]	Export the model section to the Surfer program format: data values to a DAT text file, contours to a BLN file and the color scale to an LVL file.
Section file [*.sec]	Export the current model to the internal **Zond** graphic format

	consisting of an image file and a spatial reference file in the SEC format (see the **A priori information** section for details).

Additional program features

Model smooth/raster dialog box

After editing the model or performing an inversion, several useful operations can be applied to the mesh model.

The **Model operations** module (**Options** / **Extra** / **Model smooth**/**raster**) allows you to smooth or coarsen (break up into blocks) the current model. The block model can be used for the ***Blocks*** inversion type. In this case, the model parameter is inverted for each block. Before breaking up the model into blocks, it is recommended to use a focusing inversion.

When using the **Blocks** operation in the **Model operations** module, the cells with close values of the parameter are grouped depending on the **Contrast factor** value. The **Start layer** option specifies the first layer of the range of layers this operation will be applied to.

When using the **Smooth** operation, the parameter values of neighboring cells are smoothed (averaged) depending on the **Smooth factor** value. The **End layer** option specifies the last layer of the range of layers this operation will be applied to.

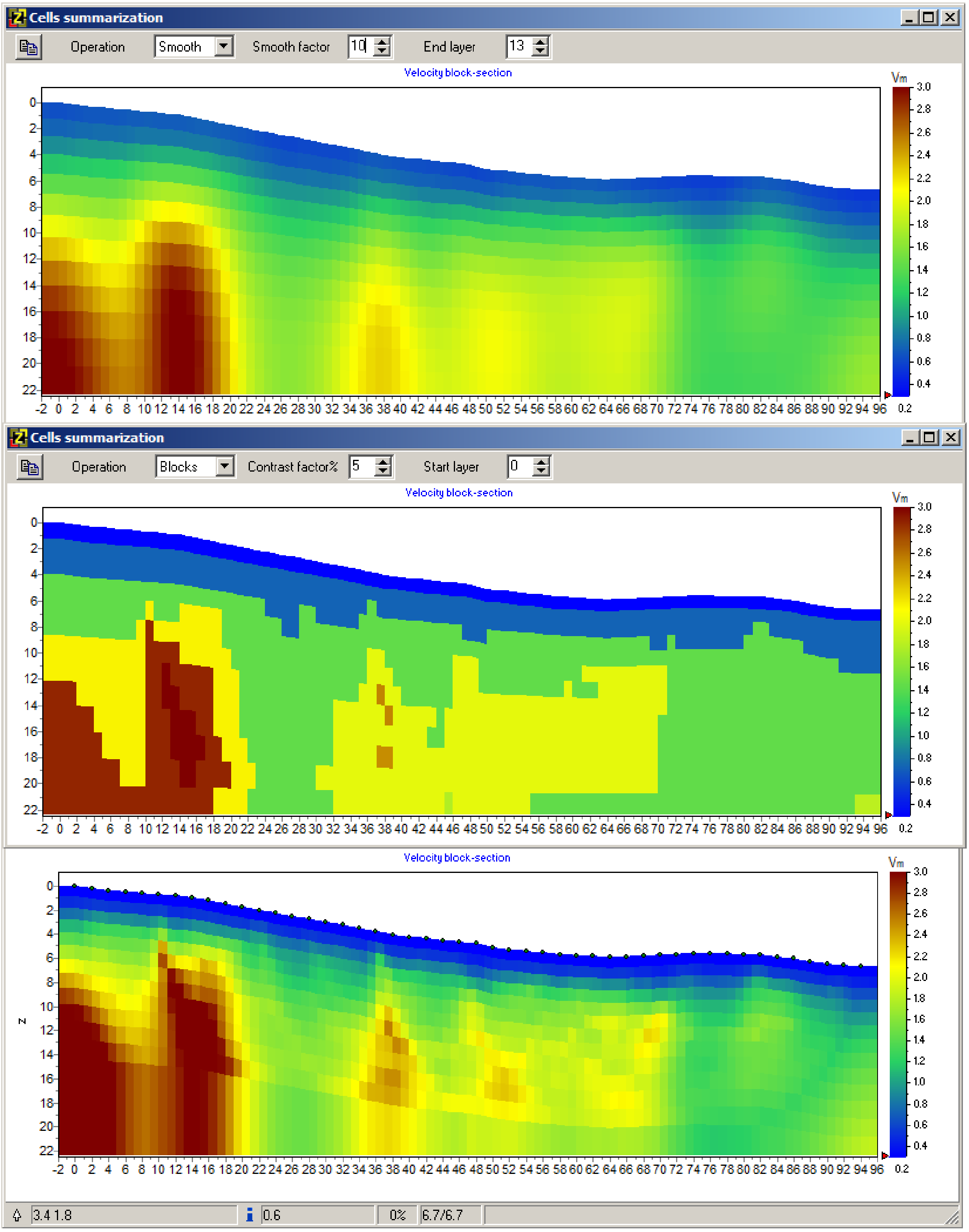
The button copies the resulting model into the model editor.

In the example below, **C** is the initial model, **B** is the model after the **Blocks** operation was applied and **A** is the model after smoothing.

A

B

C

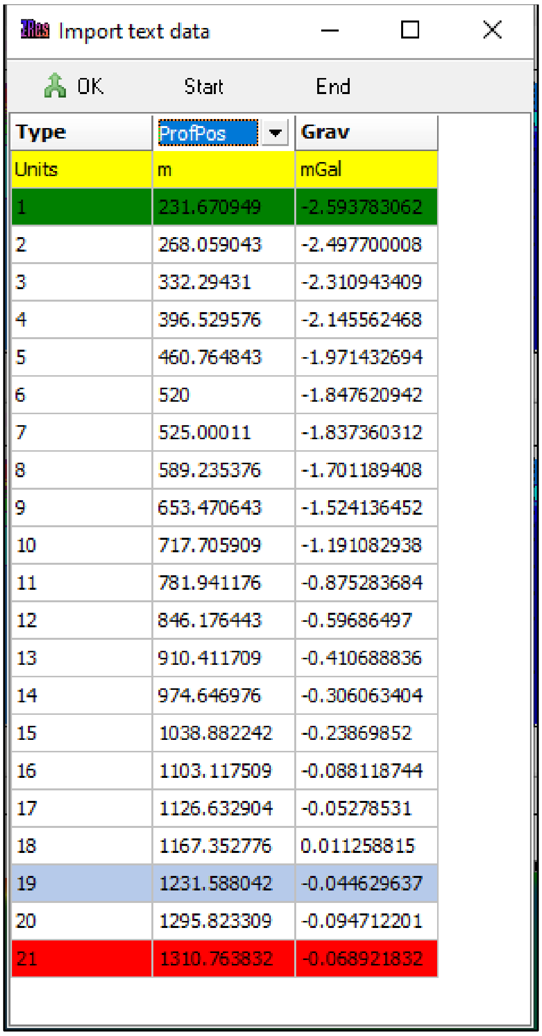


Joint inversion with gravity and magnetic data

The program can be used for joint inversion of seismic and gravimagnetic data.

For the gravity data, the program can invert measurements of the vertical component of the gravity field and its vertical gradient, for the magnetic data – measurements of the total vector and vertical gradient of the magnetic field.

Data of gravity/magnetic surveys are imported into the program from a text file or Excel spreadsheet containing two columns, with distances along the line in one column and observed values in another. To import the data, select the **Options / GraviMagnetic / Load data** menu command. An example of data being imported is provided in the figure below.

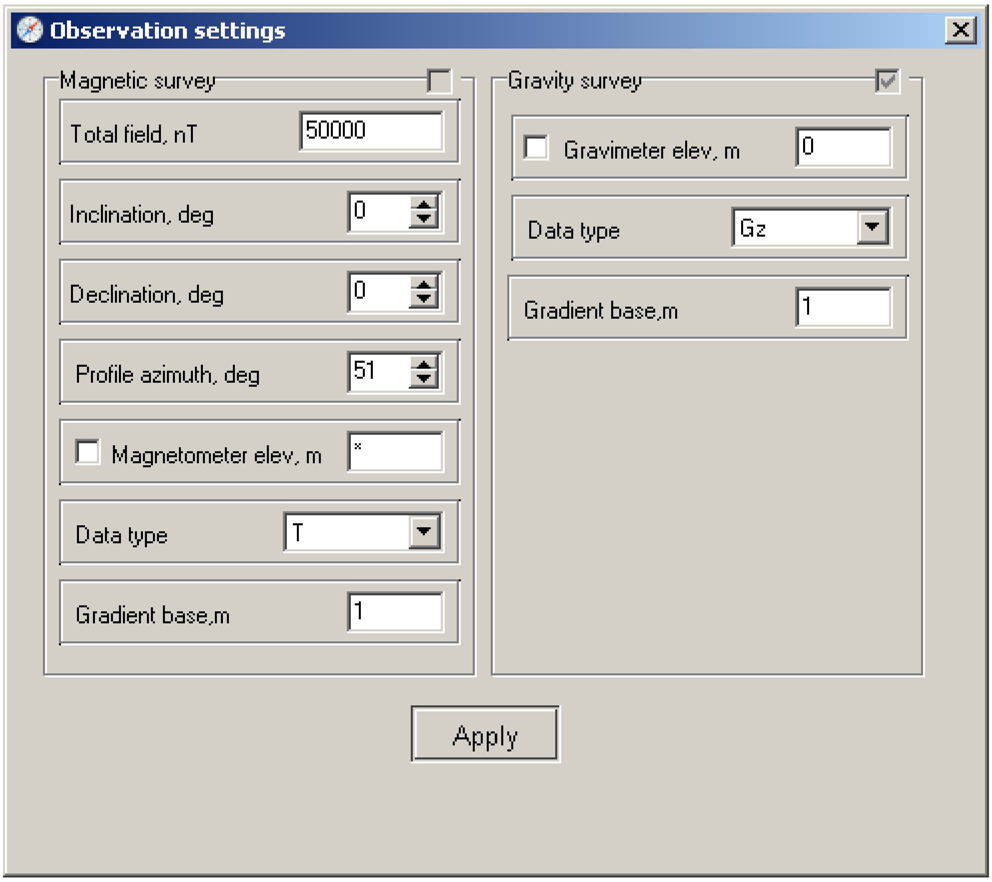


Type	ProfPos	Grav
Units	m	mGal
1	231.670949	-2.593783062
2	268.059043	-2.497700008
3	332.29431	-2.310943409
4	396.529576	-2.145562468
5	460.764843	-1.971432694
6	520	-1.847620942
7	525.00011	-1.837360312
8	589.235376	-1.701189408
9	653.470643	-1.524136452
10	717.705909	-1.191082938
11	781.941176	-0.875283684
12	846.176443	-0.59686497
13	910.411709	-0.410688836
14	974.646976	-0.306063404
15	1038.882242	-0.23869852
16	1103.117509	-0.088118744
17	1126.632904	-0.05278531
18	1167.352776	0.011258815
19	1231.588042	-0.044629637
20	1295.823309	-0.094712201
21	1310.763832	-0.068921832

If the survey data are in different files (e.g., mag data in one file and gravity data in another), you can use the **Options / GraviMagnetic / Add new data** command to append data to the project.

After selecting the file, the data import window appears (see above). Cells in the uppermost row of the table contain a drop-down list of headers: **ProfPos** (distance along the line), **Grav** (gravity data), and **Mag** (mag data). For each column, it is necessary to select the corresponding header. The second row specifies the units of measurement: for gravity data – milligals (mGal) or microgals (uGal), for magnetic data – nanoteslas (nT). Use the **Start** and **End** buttons at the top of the dialog box to specify the range of data to be imported (the **Start** button specifies the first row, the **End** button specifies the last row).

After setting the import criteria and pressing the OK button, the **Observation settings** dialog box appears. The dialog box contains settings for survey parameters and parameters of the normal magnetic field, and can also be opened at any time using the **Options / GraviMagnetic / Field settings** command.



The **Gravity survey** group box contains the following options:

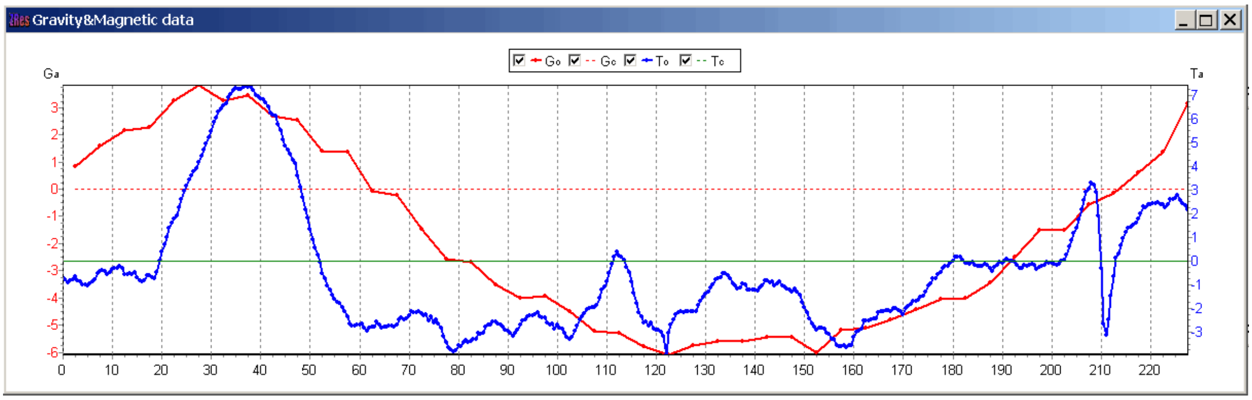
Gravimeter elev, m	Height of gravimeter above the ground (m).
Data type	Data type – vertical component of gravity field or its vertical gradient.
Gradient base, m	In the case of vertical gradient data, the measurement base (m).

The **Magnetic survey** group box contains the following options:

Total field, nT	Amplitude of the normal field (nT).

Inclination, deg	Magnetic inclination at the time of survey (degrees).
Declination, deg	Magnetic declination at the time of survey (degrees).
Azimuth, deg	Azimuth of the survey line (degrees) (from the north direction clockwise).
Magnetometer elev, m	Height of the magnetometer sensor above the ground (m).
Data type	Data type – total vector of magnetic field or its vertical gradient.
Gradient base, m	In the case of vertical gradient data, the measurement base (m) (the distance between sensors).

After setting the parameters and pressing the **Apply** button, the gravimagnetic data will appear in a new window. The window can also be opened at any time using the **Options / GraviMagnetic / Display GM window** command.



The **Options / GraviMagnetic / Substract median grav** and **Substract median mag** options perform the subtraction of the median value from the observed field, i.e., reduce the field to the anomalous component.

After creating a polygonal model in the model editor, you can either calculate the forward gravity/magnetics problem (perform modeling in the polygonal mode) or perform the data inversion. Double-clicking on the gravimagnetic data window recalculates the forward problem for a current polygonal model. To start the inversion of density and/or magnetic susceptibility for a given framework of polygons, use the **Options / GraviMagnetic / Inversion** command.

To set the density and/or magnetic susceptibility of a polygonal object, use the **Body parameters** dialog box (see the **Polygonal model mode** section for details).

In the layered model mode, the density or magnetic susceptibility can be set by selecting the **Gravity** or **Magnetic** option in the model constructor settings dialog box **(Layered / Model constructor)**.

Joint inversion can be performed in all three modeling modes:

Block (mesh) mode. A joint inversion of ST data with gravimagnetic data is performed. The **Options / Inversion / Cross-gradient** menu subsection is used.

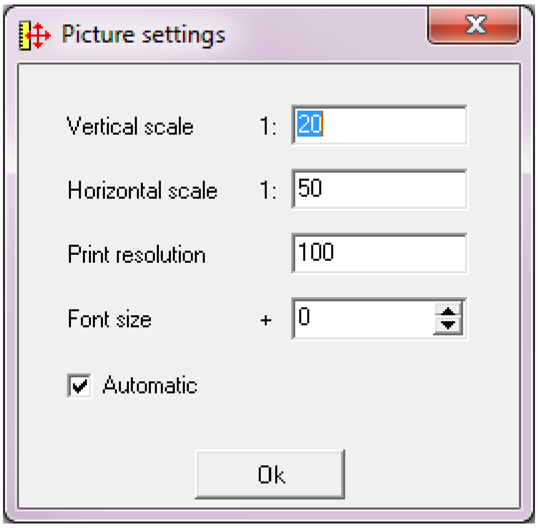
Polygonal mode. The inversion is performed independently for ST data and gravimagnetic data within the same polygonal model framework. The **Options / GraviMagnetic / Inversion** command is used.

Layered mode. A joint inversion of ST data with gravimagnetic data is performed based on the common geometry of boundaries. The **Options / GraviMagnetic / Invert gravity** and/or **Options / GraviMagnetic / Invert magnetic** options are used.

Graphic Settings

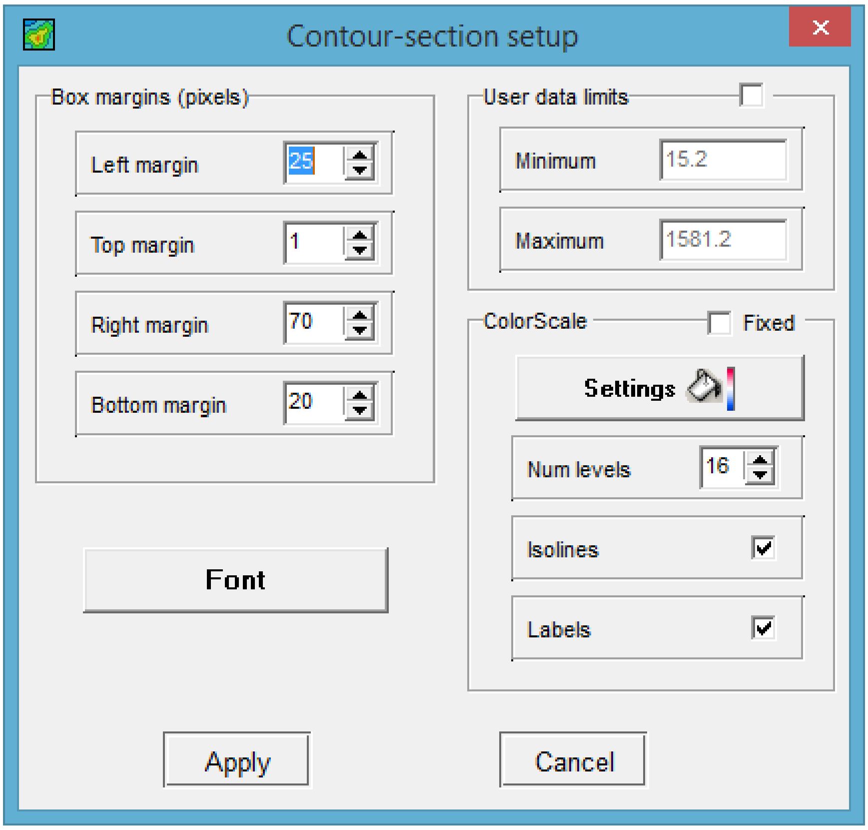
Export image settings

The **Picture settings** dialog box is opened with the **Options / Graphics / Bitmap output settings** command. With the **Automatic** option disabled, you can change the **Vertical scale**, **Horizontal scale**, **Print resolution** (DPI) and **Font size** of the export image.



Contour section and pseudosection settings

To open the **Contour-section setup** dialog box, right-click on the title area of the model contour section or on the color scale and select the **Setup** option in the context menu.



The dialog is used to configure the contour section properties.

The **Box margins (pixels)** group box contains the following options:

Left margin – sets the margin area (in pixels) on the left side of the image.

Right margin – sets the margin area on the right side of the image.

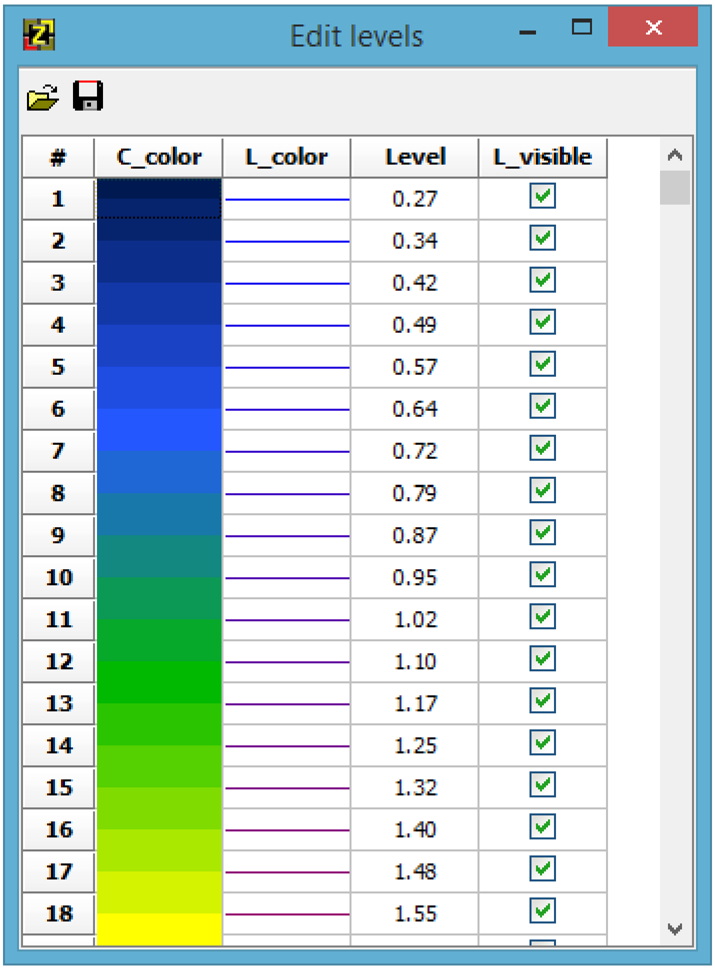
Top margin – sets the margin area on the top of the image.

Bottom margin – sets the margin area on the bottom of the image.

If the **User data limits** option is enabled, the values in the **Minimum** and **Maximum** input boxes are used to set the range of values for the contour plot. If disabled, the data limits are used.

The **ColorScale** group box contains the following options:

Settings – sets the color palette.



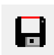
Right-click on the table headers performs the following actions:

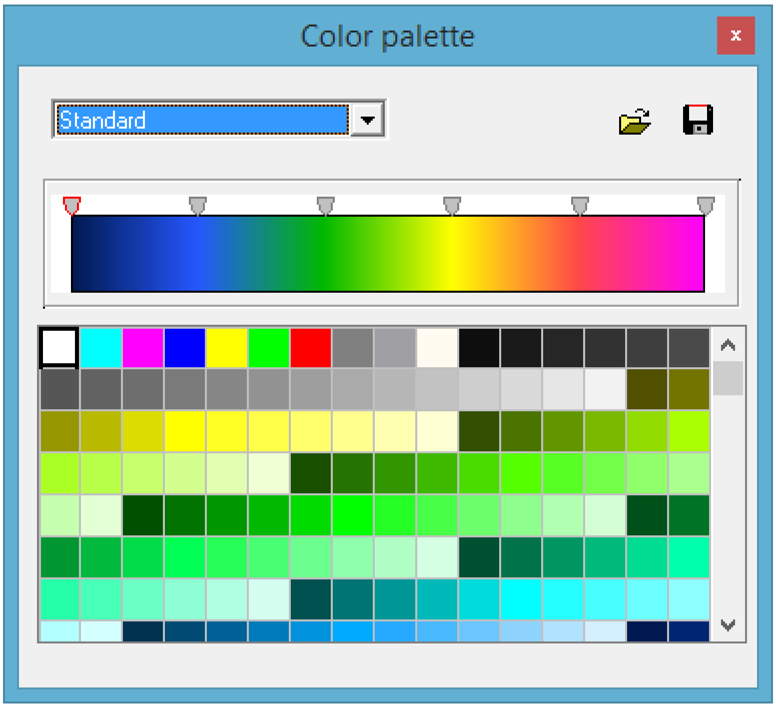
C_color – opens the dialog box for editing the fill color palette.

L_color – opens the dialog box for editing the line color palette.

Level – opens the dialog box for specifying the range of values.

L_visible – uncheck/check all boxes.

The **Color palette** dialog box allows you to choose a preset palette or create a custom one. To add a slider to the palette, left-click on the slider area while holding down the Ctrl key. To delete a slider, select it and press the Delete key. Custom palettes can be saved and loaded using the  and  buttons.



You can save and load color palettes in the Surfer color scale format (CLR).

Num levels – defines the number of contour levels. The contour level method can be set to linear or logarithmic, depending on the data type.

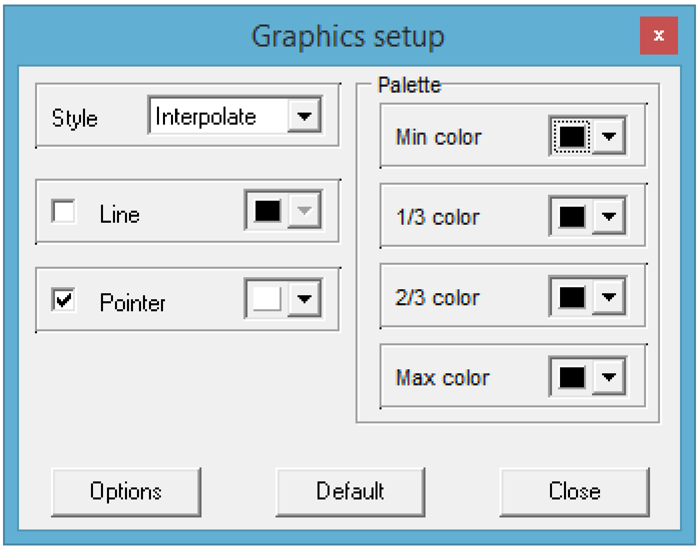
Isolines – shows or hides isolines.

Labels – shows or hides isoline labels.

The **Font** button opens a dialog box for changing the color scale font.

Graphic plot settings

To open the dialog box for changing properties of a graphic plot (a set of graphs) displayed in the main program window, select the **Options** / **Graphics / Observed graphics** or **Calculated graphics** menu command.



Style – defines the method which is used to assign colors to a set of graphs.

Interpolate – a color palette is created by interpolating the set of colors specified in the **Palette** group box (*Min color*, *1/3 color*, *2/3 color* and *Max color*).

Constant – the same color is assigned to all graphs.

Random – random colors are assigned to graphs.

Line – specifies the connecting line color. If the checkbox is checked, the color set in the **Line** box is assigned to lines, otherwise, colors from the **Palette** group box are used.

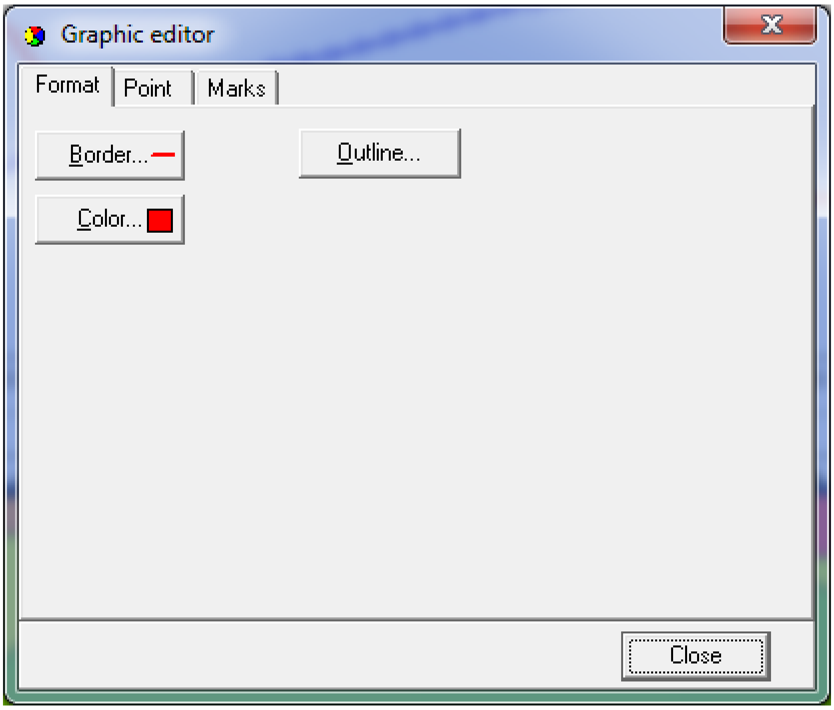
Pointer – specifies the graph point fill color. If the checkbox is checked, the points are filled with the color set in the **Pointer** box, otherwise, colors from the **Palette** group box are used.

The **Default** button restores the default dialog box settings.

The **Options** button opens the **Graphic editor** dialog box.

Graph settings

The **Graphic editor** is intended to customize the appearance of graphs. For graphic plots, it is accessible through the **Graphics setup** dialog box (the **Options** button). In the case of individual graphs, it can be opened by right-clicking on the graph while holding down the Shift key.



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

The **Border** button opens the dialog box for configuring the connecting lines.

The **Color** button opens the line color settings dialog box.

The **Outline** button opens the stroke outline settings dialog box.

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

The **Visible** checkbox shows or hides the graph points.

The **Error gates** checkbox shows or hides confidence intervals.

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

Width – sets the point width in pixels.

Height – sets the point height in pixels.

The **Pattern** button opens the point fill settings dialog box.

The **Border** button opens the point outline settings dialog box.

The **Marks** tab contains settings for the graph point labels.

The **Style** sub tab contains the following settings:

The **Visible** checkbox shows or hides the point labels.

Draw every – allows plotting every second, third, etc. label depending on the selected value.

Angle – determines the orientation angle of the point labels.

If the **Clipped** checkbox is activated, the labels that go outside the graph area are not plotted.

The **Arrows** sub tab contains settings customizing the appearance of arrows that go from the labels to graph points.

The **Border** button opens the arrow line settings dialog box.

Length – specifies the length of the arrows.

Distance – sets the distance between the arrowhead and the graph point.

The **Format** sub tab contains the label graphic settings.

The **Color** button opens the dialog box for specifying the label's background color.

The **Frame** button opens the frame line settings dialog box.

Round Frame – plots the rounded corner frame.

Transparent – sets the transparency of the label's background.

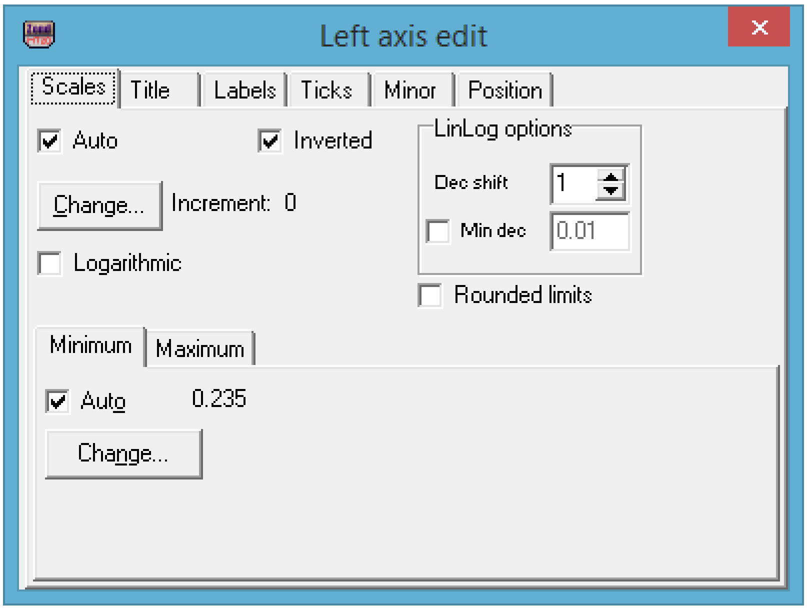
The **Text** sub tab contains the following settings:

The **Font** button opens the dialog box for the label font settings.

The **Outline** button opens the dialog box for the label text outline settings.

Axis editor

The **Axis editor** is used to change axes configuration. It can be opened by right-clicking on the axis of interest while holding down the Shift key. A context menu will appear with three items: **Options**, **Default** and **Fix range**. The first option opens the **Axis editor** dialog box, the second sets the axis configuration parameters to default values.



The **Scales** tab of the dialog box contains axis scaling options.

If the **Auto** checkbox is checked, the program will determine the minimum and maximum of the axis automatically. Otherwise, the values set by the user in the **Minimum** and **Maximum** sub tabs are used.

The **Inverted** checkbox determines the axis direction.

The **Increment Change** button opens the dialog box for specifying the label interval.

The **Logarithmic** checkbox sets the axis scale to logarithmic or linear. If negative or zero values have to be represented, you should additionally use the options in the **LinLog options** group box.

The **LinLog options** group box contains options for configuring the linear-logarithmic axis. The linear-logarithmic scale allows to represent negative or zero values on a logarithmic scale.

Dec Shift – sets the offset in decades from the maximum axis limit (by absolute value) to zero. The decade closest to zero will have a linear scale, the rest of the axis will be logarithmic.

Min dec – sets the value of the decade closest to zero when the checkbox is checked.

If the **Rounded limits** checkbox is activated, the program will round the axis minimum and maximum values.

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

If the **Auto** checkbox is activated, the axis limit is determined automatically. Otherwise, the axis limit can be set by pressing the **Change** button.

The **Title** tab contains axis title configuration options.

The **Style** sub tab contains the following options:

Title – defines the text for the axis title.

Angle – determines the orientation angle of the axis title.

Size – specifies the title offset. If set to 0, the offset is determined automatically.

The **Visible** checkbox shows or hides the axis title.

The **Text** sub tab contains the following options:

The **Font** button opens the font settings dialog box.

The **Outline** button opens the text outline settings dialog box.

The **Labels** tab contains axis labels configuration options.

The **Style** sub tab contains the following options:

The **Visible** checkbox shows or hides the axis labels.

Offset – specifies the label offset. If set to 0, the offset is determined automatically.

Angle – determines the orientation angle of the axis labels.

Min. Separation % – specifies the minimum separation (in percentage) between labels.

The **Text** sub tab contains the following options:

The **Font** button opens the font settings dialog box.

The **Outline** button opens the text outline settings dialog box.

The **Ticks** tab contains axis tick marks configuration options.

The **Axis** button opens the axis line settings dialog box.

The **Grid** button opens the major grid lines settings dialog box.

The **Ticks** button opens the dialog box for configuring the major external tick marks. The **Len** option specifies their length.

The **Inner** button opens the dialog box for configuring the major internal tick marks. The **Len** option specifies their length.

If the **At labels only** checkbox is activated, the tick marks will be plotted only where labels are present.

The **Minor** tab contains minor tick marks configuration options.

The **Ticks** button opens the dialog box for configuring the minor external tick marks.

The **Grid** button opens the minor grid lines settings dialog box.

Length – sets the length of minor tick marks.

Count – sets the number of minor tick marks between the major tick marks.

The **Position** tab contains options for specifying the dimensions and position of the axis.

Position % – sets the axis offset on the graph relative to the default position (as a percentage of the total graph size).

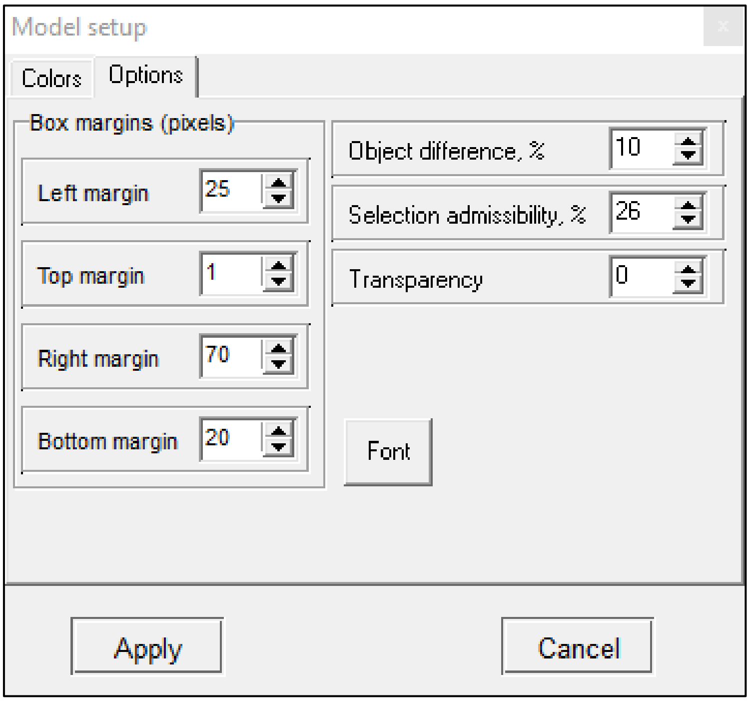
Start % – sets the offset of the start of the axis on the graph relative to the default position (as a percentage of the total graph size).

End % – sets the offset of the end of the axis on the graph relative to the default position (as a percentage of the total graph size).

The **Other side** checkbox plots the axis on the opposite side of the graph. For example, if the default axis position is at the bottom, checking this option will position the axis at the top.

Model section settings

To open the **Model setup** dialog box, right-click on the title area of the model section or on the color scale (when not in the *Contour-section* display mode) and select the **Setup** option in the context menu.



The **Options** tab of the dialog box contains the following options:

The **Box margins (pixels)** group box specifies the model section margins.

Left margin – sets the margin area (in pixels) on the left side of the image.

Right margin – sets the margin area on the right side of the image.

Top margin – sets the margin area on the top of the image.

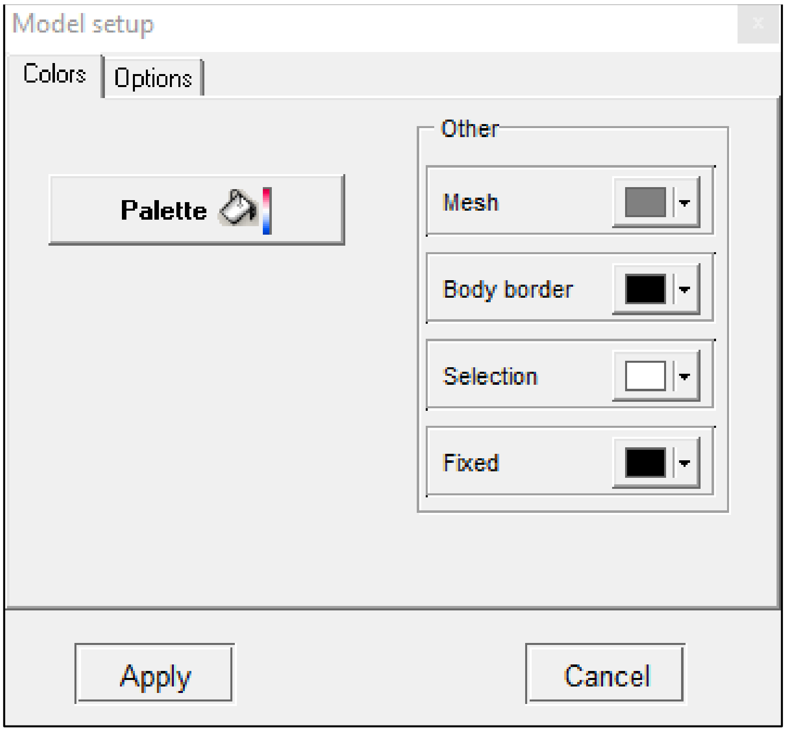
Bottom margin – sets the margin area on the bottom of the image.

Object difference, % – sets the ratio (in percentage) between parameter values of adjacent cells, above which a boundary is plotted between the cells.

Selection admissibility, % – sets the degree of similarity for adjacent cells to be selected as a single object with the “Magic wand” selection tool.

Transparency – sets the transparency of the background image.

The **Font** button opens the font settings dialog box.



The **Colors** tab contains the following options:

The **Palette** button opens the dialog box for configuring the color palette.

Mesh – specifies the mesh color.

Body border – specifies the color of the boundary between adjacent cells.

Selection – specifies the color of the selection fill pattern.

Fixed – specifies the color of the fill pattern of fixed cells.

Print preview

The **Print preview** window is opened with the **File / Print preview** menu command.

You can move objects on the sheet by dragging them with the mouse.

The toolbar of the **Print Preview** window contains the following items:

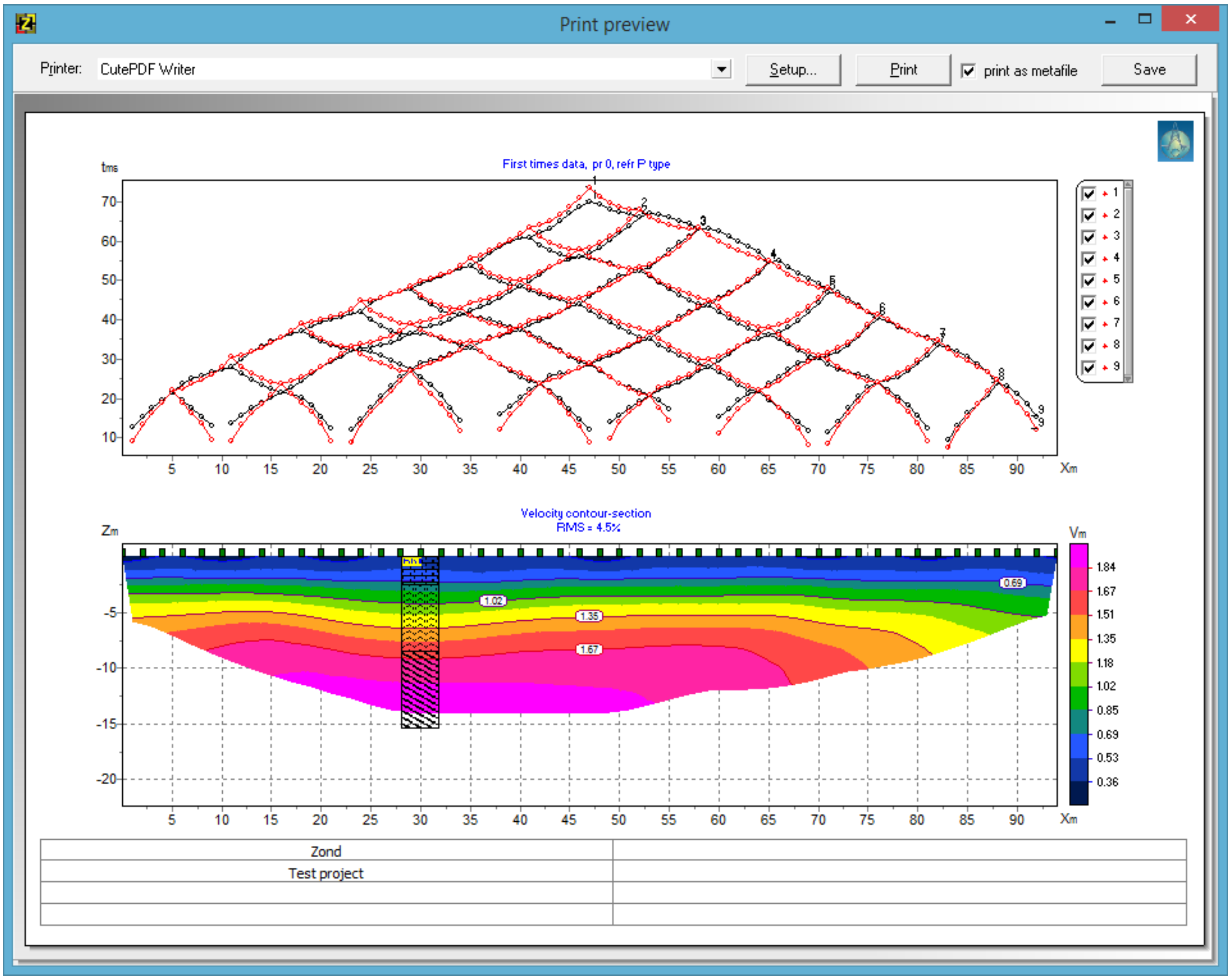
– select a printer.

– open a standard print setup dialog box.

– print the drawing.

– if activated, the drawing is saved as a vector image.

– save as a bitmap image.



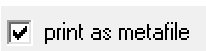
Use the left mouse button to move the print object on the sheet.

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

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

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

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



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

- saving to bitmap files.

The square area in the top left corner of the sheet can be used for seals, stamps or company logos. Right-click on the area to upload a BMP image. The area can be resized with the mouse.

At the bottom of the sheet is an editable table. Right-click on the table for text input. The contents of the table can be saved and loaded using the and buttons.

Additional information

Youtube video lessons

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

LinkedIn support group

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

Demo Zond projects

ftp://zond-geo.com/

Username: download@zond-geo.com

Password: 12345

USB dongle troubleshooting

If the program does not work with a USB dongle:

1. The dongle driver is not installed or has not been installed correctly. In some operating systems the dongle is recognized as an HID device and there is no need to install a driver, but in some systems, it is not recognized correctly and the driver needs to be installed. Driver download link: *http://senselock.ru/files/senselock_windows_3.1.0.0.zip*. The dongle should appear in the device manager as "Senselock Elite".

2. The period of free updates has ended. In this case, you have to use the latest working version of the program or purchase additional 2 years of updates.

3. Sometimes, when the dongle turns into HID mode, the system might not recognize it as an HID device. In this case, you need to switch it back to USB mode using a small application that can be downloaded from the following link: *http://www.zond-geo.com/zfiles/raznoe/SenseSwitch.zip*. The **senseswitch.exe** has to be started from the command prompt using the command **senseswitch.exe usb**.