GEODEVICE
GEOPHYSICS

OPERATION MANUAL

GeoTechControl

CONTENTS

INTRODUCTION

Pile-MASTER software is designed for processing and interpretation of the data obtained by the seismoacoustic method of surveying building structures.

Software Pile-MASTER is applied when studying:
- driven and bored piles;
- foundation reinforced concrete slabs;
- tunnels.

Software Pile-MASTER allows to:

1. process the data obtained by Sonic Integrity Testing and the impact method in order to solve the following problems:
    a. determination of piles' length,
    b. determination of pile integrity,
2. conduct dynamic attribute analysis of seismic gathers for:
    a. searching for weakened piles with reduced concrete strength,
    b. evaluations of pile-soil contact conditions,
    c. assessment of the condition of the foundation slab with ground,
    d. search for voids behind tunnel lining.
3. solve other geotechnical problems related to the processing of seismoacoustic data.

1 PROGRAM SETUP AND RUNNING

To install the program run the file "setup.exe" and then follow the instructions.

The following points should be kept in mind during setup and operation of the program:

1. To successfully install the GeoTechControl software, user needs administrator rights.

2. Once the software is successfully installed, GeoTechControl can be launched either through the Windows Start menu or through a shortcut created on the desktop.

3. When installing a new version of GeoTechControl software on a computer that already has one of the previous versions installed, you do not need to uninstall the already installed version. The new version will automatically overwrite the old one.

4. When installing an older version of GeoTechControl software on a computer on which a later version of the software is already installed, it is recommended that you uninstall the current version before installing the older one.

5. After successful installation, a working folder of the program will be created, which can be found in Windows Explorer at: "%appdata%/GeoTechControl».

    a. The folder «%appdata%/ GeoTechControl /Data» is recommended for storing projects as well as raw and processed data.

    b. Program logs are stored in the folder «%appdata%/GeoTechControl/Logs». If errors are detected, technical support will request data from this folder.

System requirements

- Recommended operating systems: Windows 10, Windows 7.
- Preinstalled ".NET Framework" version 4.7.2 or higher.
- RAM: at least 2 Gb.
- Intel(R) Core(TM) i3 or better processor.

2 WORKING WITH PROJECTS

2.1 GeoTechControl main window

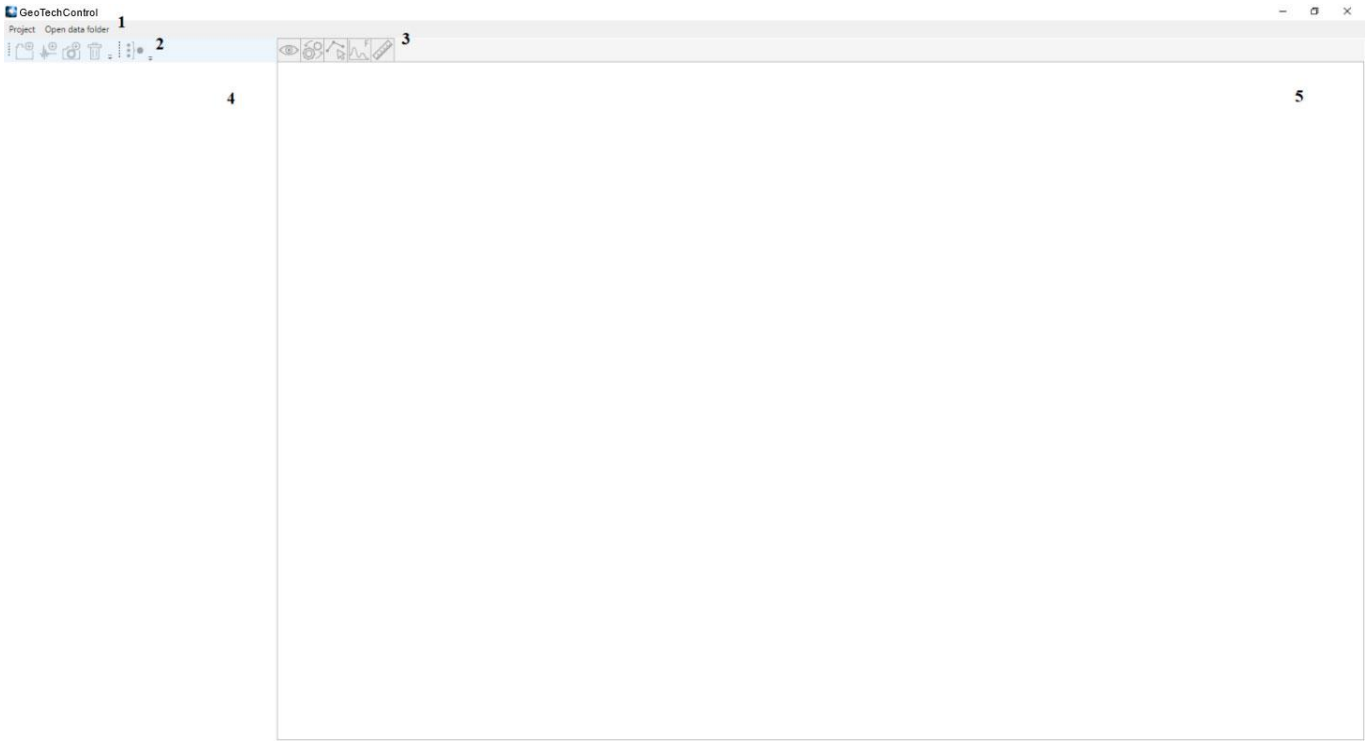

Fig. 1. Program main window

Main window of the GeoTechControl program (Fig. 1) have the following functions:

1. Main menu
   a. Project management
   b. Opening the Impact Method window
   c. Downloading a graphical report
   d. Opening the working folder of the program
2. Project toolbar
   a. Add a folder to the project
   b. Add a seismic gather to the project
   c. Add an image to the project
   d. Assembly of seismic gathers
3. Open file toolbar
   a. Signal display parameters
   b. Signal processing parameters
   c. Trace picking
   d. Signal spectrum
   e. Interpretation panel
4. Project tree
5. Seismic gather / image display area

2.2 Project creation

A project in the GeoTechControl software is a local SQLite database file with the ".db" extension. The database stores all files that are added to the project, as well as the display, processing and interpretation settings for each file.
To create a project, select the "Project - New Project" menu item (Fig. 2).

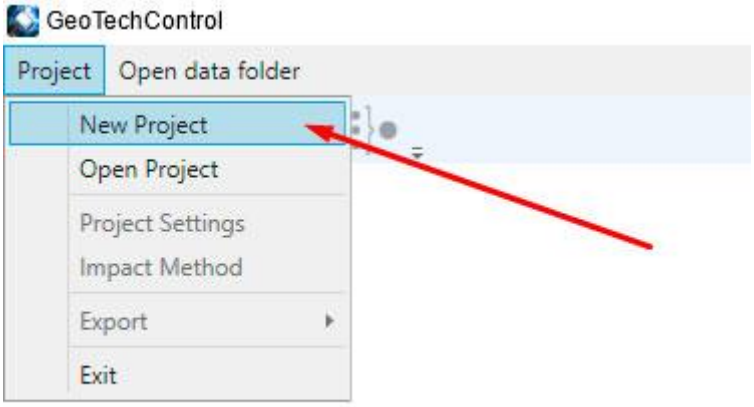

Fig. 2. Menu choice for new project creation

The "Create Project" dialog box will open. Enter the name of the new project (without extension) and click "Save" (Fig. 3). To cancel the procedure for creating a new project, click "Cancel".

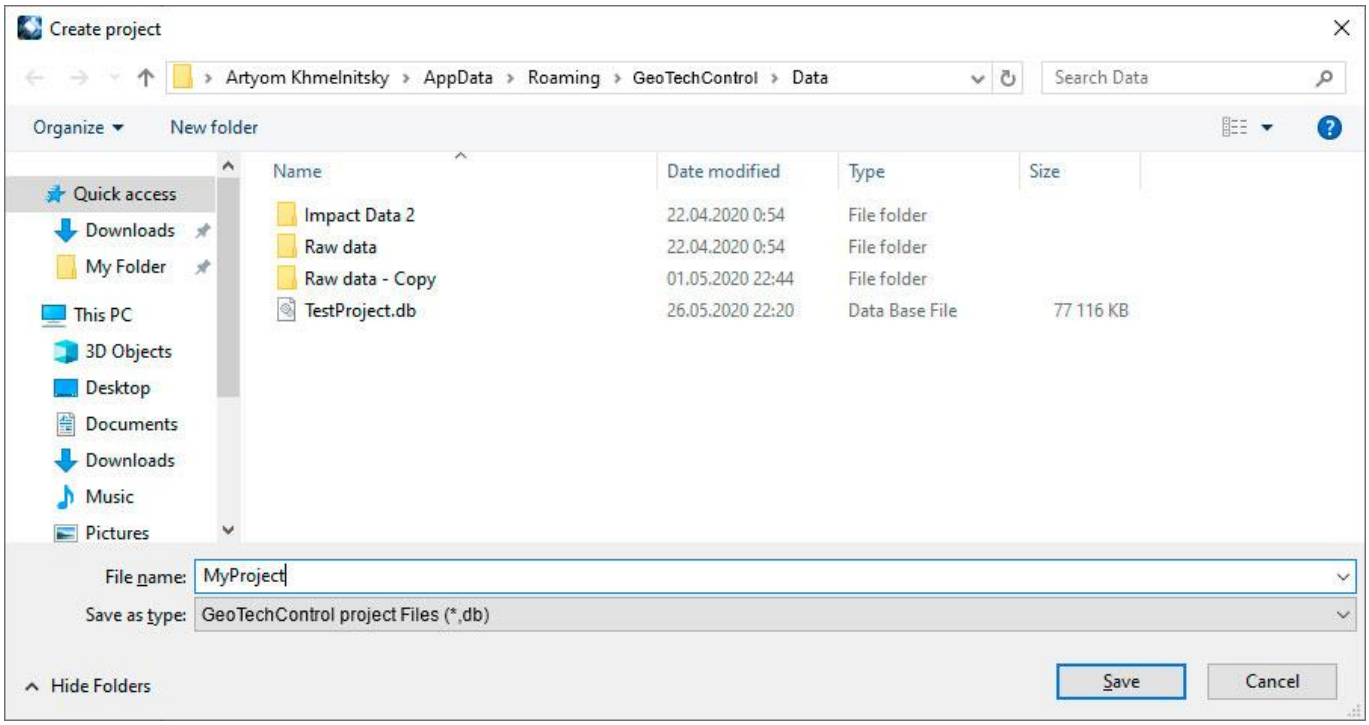

Fig. 3. Project creation dialog box

After clicking "Save" in the appearing dialog box, choose whether or not to create a subfolder for the project (Fig. 4).

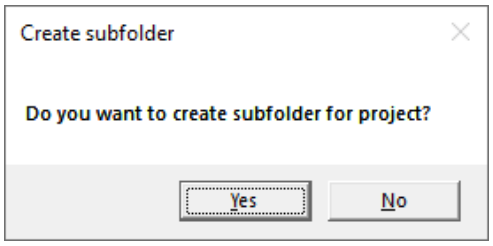

Fig. 4. Project subfolder creation dialog box

If you chose not to create a subfolder, the database file will appear in the folder selected in the "Create Project" dialog box (Fig. 5).

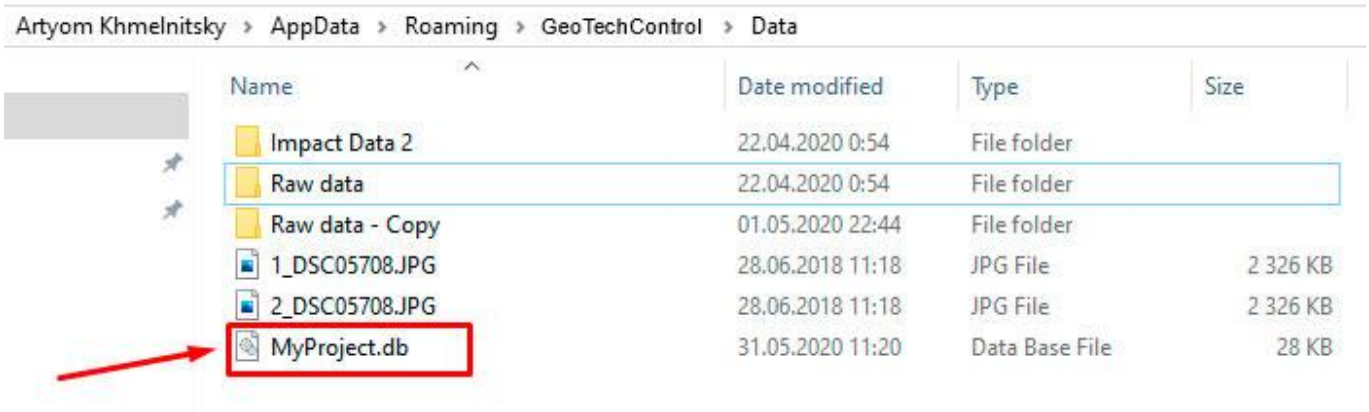

Fig. 5. New project file without creating a subfolder

If the choice to create a subfolder was made, in the folder selected in the "Create Project" dialog box, a folder with the name of the new project will be created. The database file will appear in the new folder with the project name (Fig. 6).

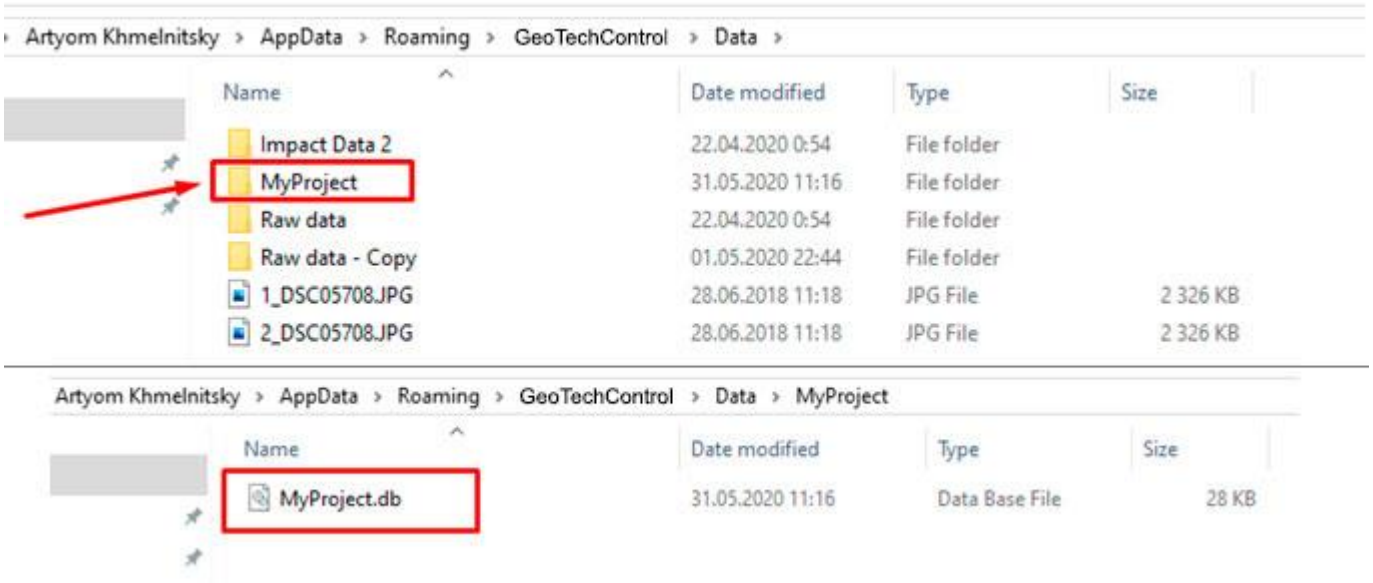

Fig. 6. New project file with subfolder creation

After successful creation, the project will be opened in the program. The program header and the project tree will show the project name (Fig. 7).

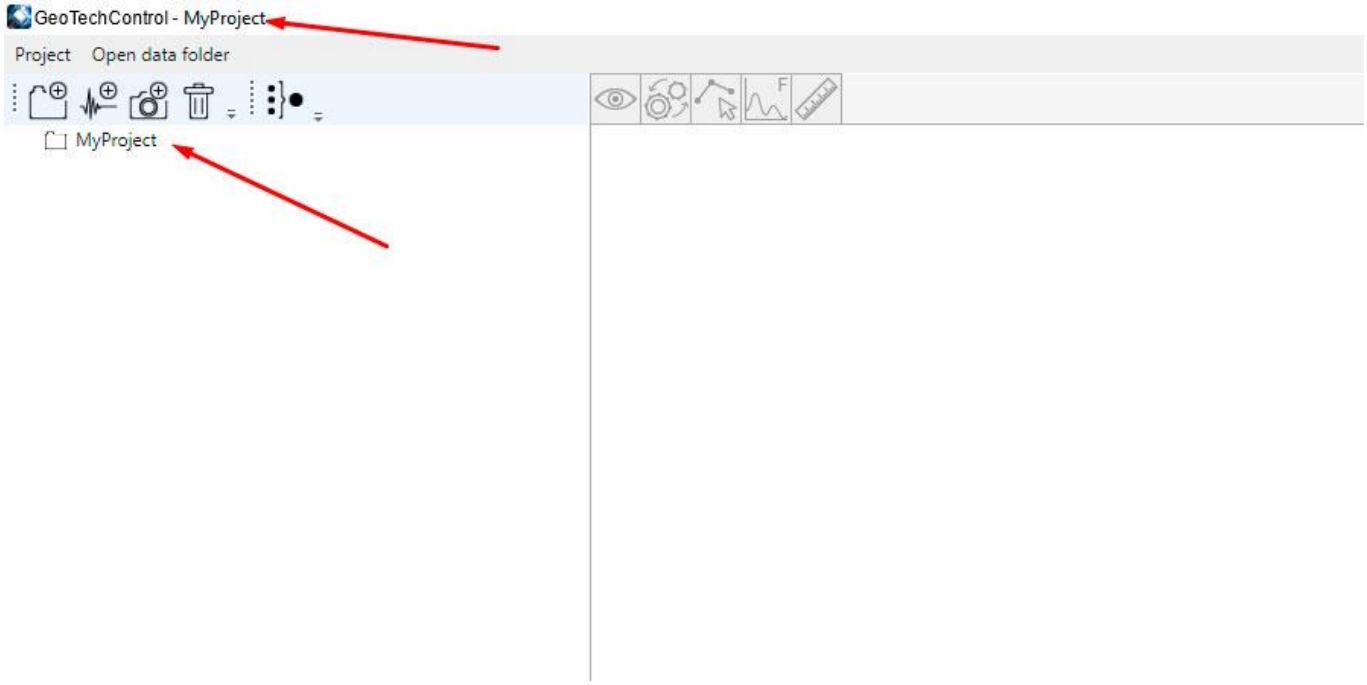

Fig. 7. Project opened in the program

2.3 Project opening

An existing project can be opened through the menu "Project – Open Project" (Fig. 8).

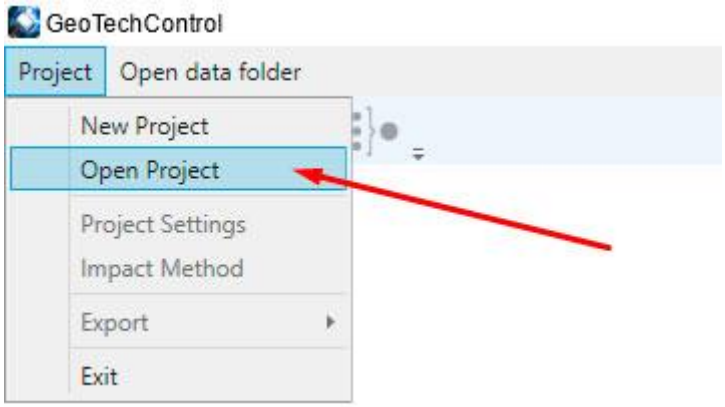

Fig. 8. Project opening menu

After selecting "Open Project" the project file selection dialog box will open. Select the desired project file and click "Open" (Fig. 9).

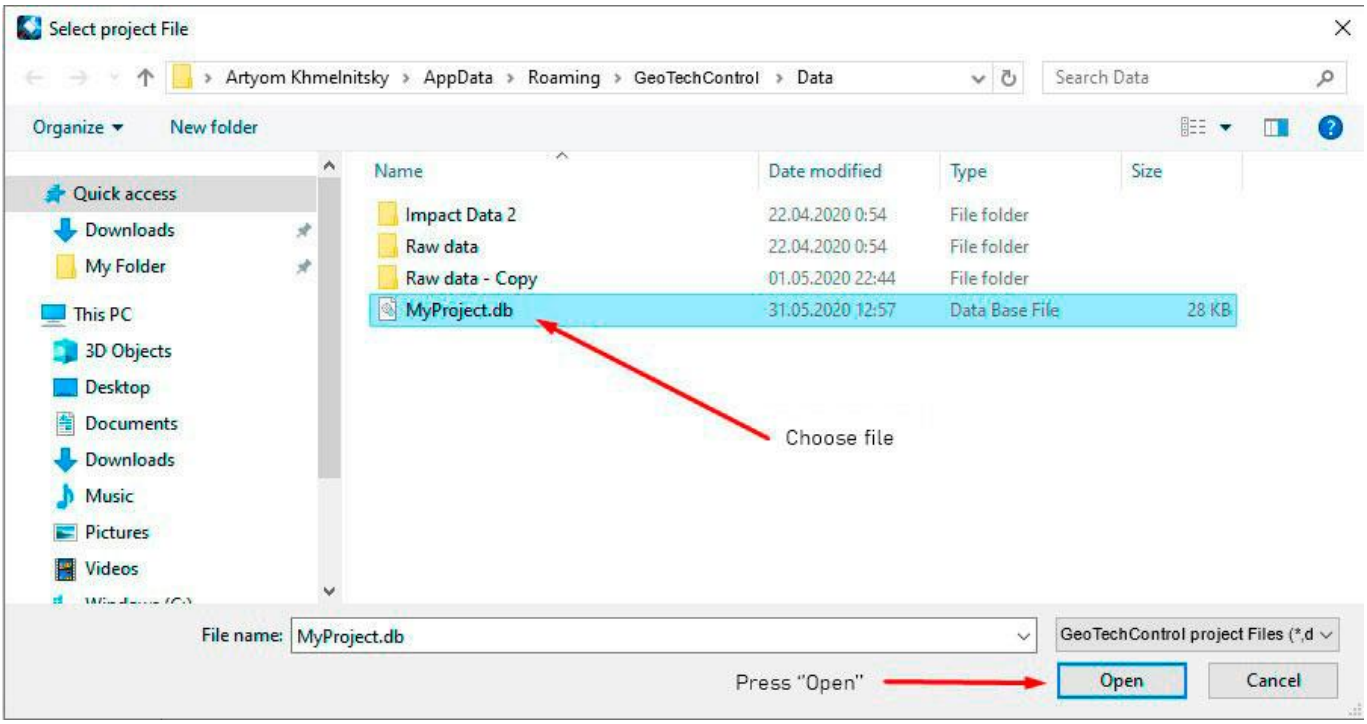

Fig. 9. Open project file dialog window

The selected project will be opened in the program (Fig. 7).

To transfer a GeoTechControl project to another computer, simply transfer the database file and open it via the "Project - Open Project" menu. All the data loaded into the project will be transferred in the project file "db".

2.4 Project tree

Working with the project tree is similar to working with folders and files in the Windows file browser (Explorer). It is possible to organize data storage in the project in folders.
A hierarchical folder structure with no nesting restrictions is supported.

Two types of files can be added to folders:
1. Seismic gathers
   a. SegY or ddb files, recorded by 2-channel seismograph IDS-1 (Logis-Geotech, Russia).
   b. Tfl files, created by Pile-MASTER software.
2. Images in JPEG format.

2.4.1. Adding folders to a project

To add a folder to the project, select a folder already existing in the project and click the "Add folder" button on the toolbar (Fig. 10).

Fig. 10. Adding folder to a project

Important: New folders can only be added to existing ones. If when you click the "Add folder" button no folder is selected in the project tree, an error window will appear (Fig. 11).

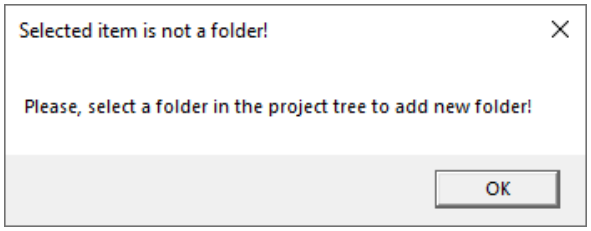

Fig. 11. Error message when adding a folder to the project

If when you click the "Add folder" button in the project tree a folder was selected, the window for entering a new folder name appears (Fig. 12).

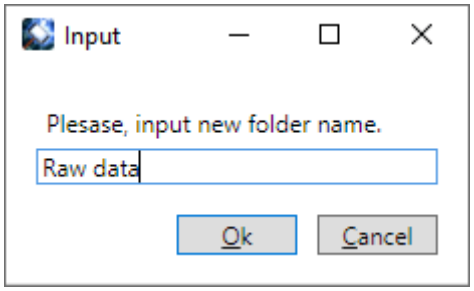

Fig. 12. New folder name dialog box

Enter a name and the new folder will be added to the project. In order to see the new folder expand the one that was selected when adding the new folder (Fig. 13).

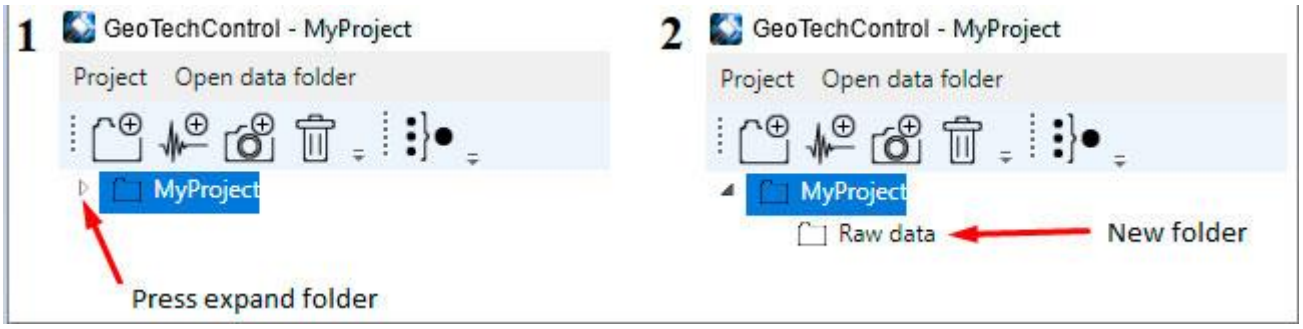

Fig. 13. Expanding a folder in the project tree

2.4.2. Adding seismic gathers to a project

To add a seismic gather to the project, select the folder already existing in the project and click the button "Add seismic files" on the toolbar (Fig. 14).

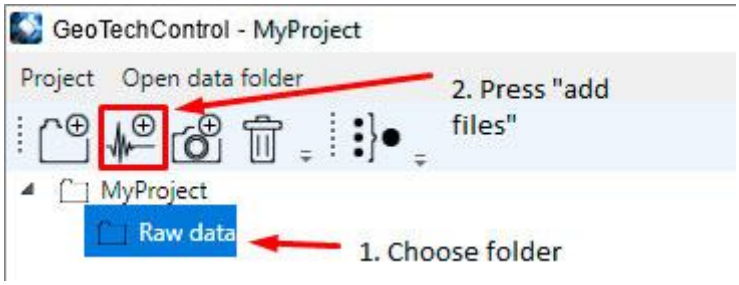

Fig. 14. Adding seismic gather to a project

Important: seismic gathers can be added only to the already existing folders. If when clicking the button "Add seismic files" the folder in the project tree is not selected, the window with an error message will appear (Fig. *15*).

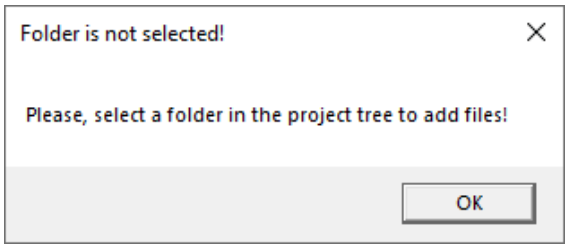

Fig. 15. Error message when adding seismic gather to a project

If by clicking the button "Add seismic files" a folder was selected in the project tree, the dialog box of file selection will appear. Select one or more seismic files and click "Open" (Fig. 16).

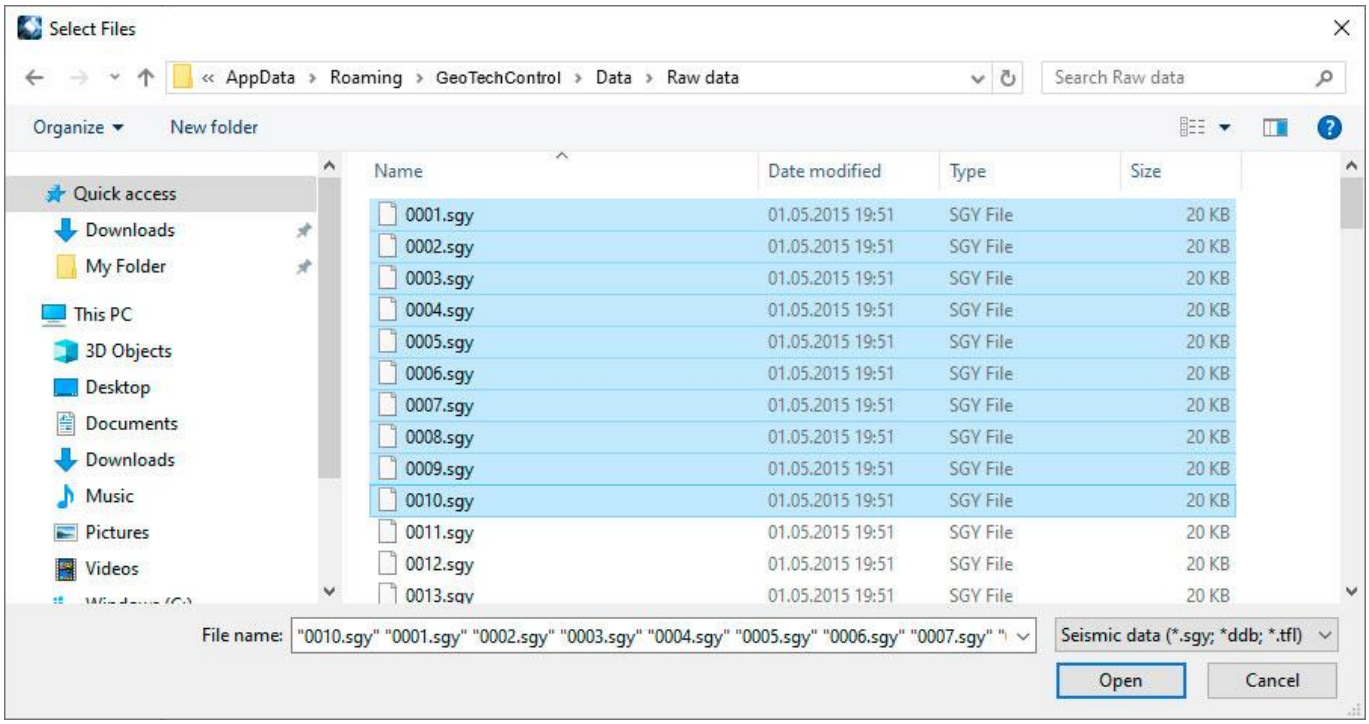

Fig. 16. Seismic gather file selection dialog box

In order to see the files added to the project, expand the folder that was selected before clicking "Add seismic files" (Fig. 17).

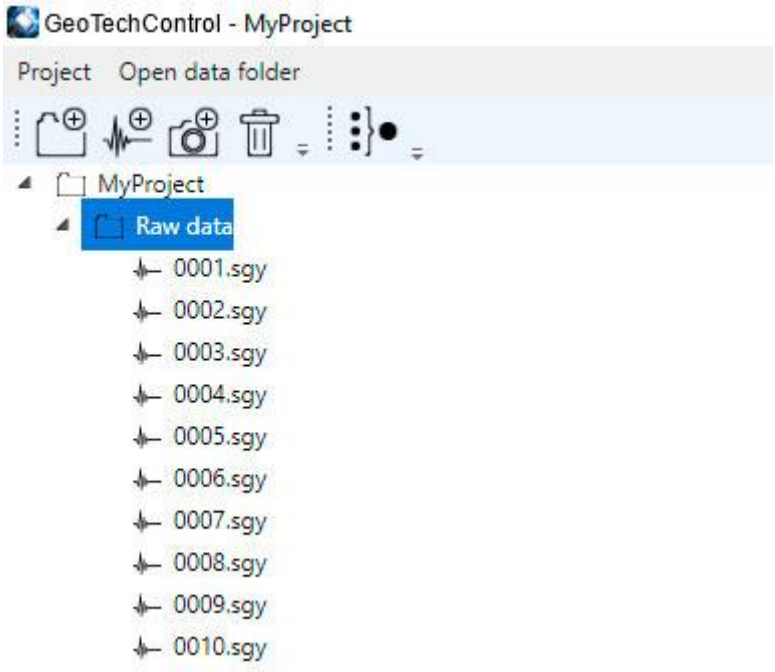

Fig. 17. New files added to the project tree

2.4.3. Adding photos to a project

To add photos to the project, select a folder already existing in the project and click the "Add photos" button on the toolbar (Fig. 18).

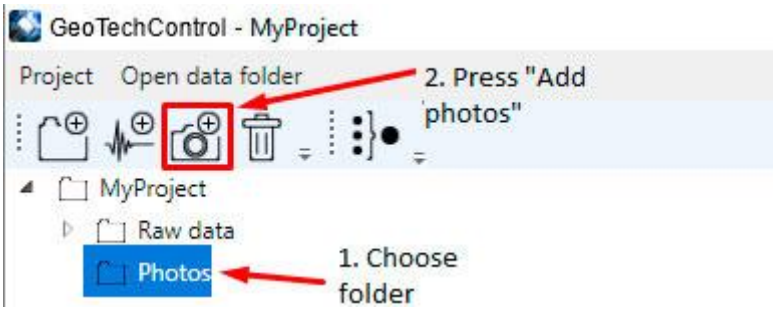

Fig. 18. Adding photo to a project

Important: Seismic gathers can be added only to the already existing folders. If the folder in the project tree is not selected when clicking the button "Add photos", a window with an error message will appear (Fig. 19).

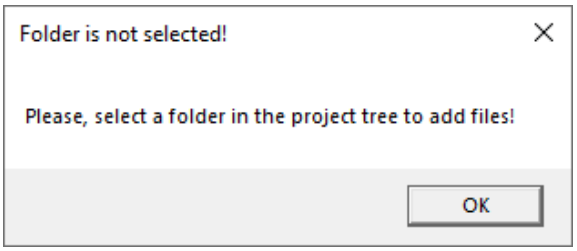

Fig. 19. Error message when adding seismic gathers to a project

If a folder has been selected in the project tree when you click the "Add photos" button, a file selection dialog box will appear. Select one or more photos and click "Open" (Fig. 20).

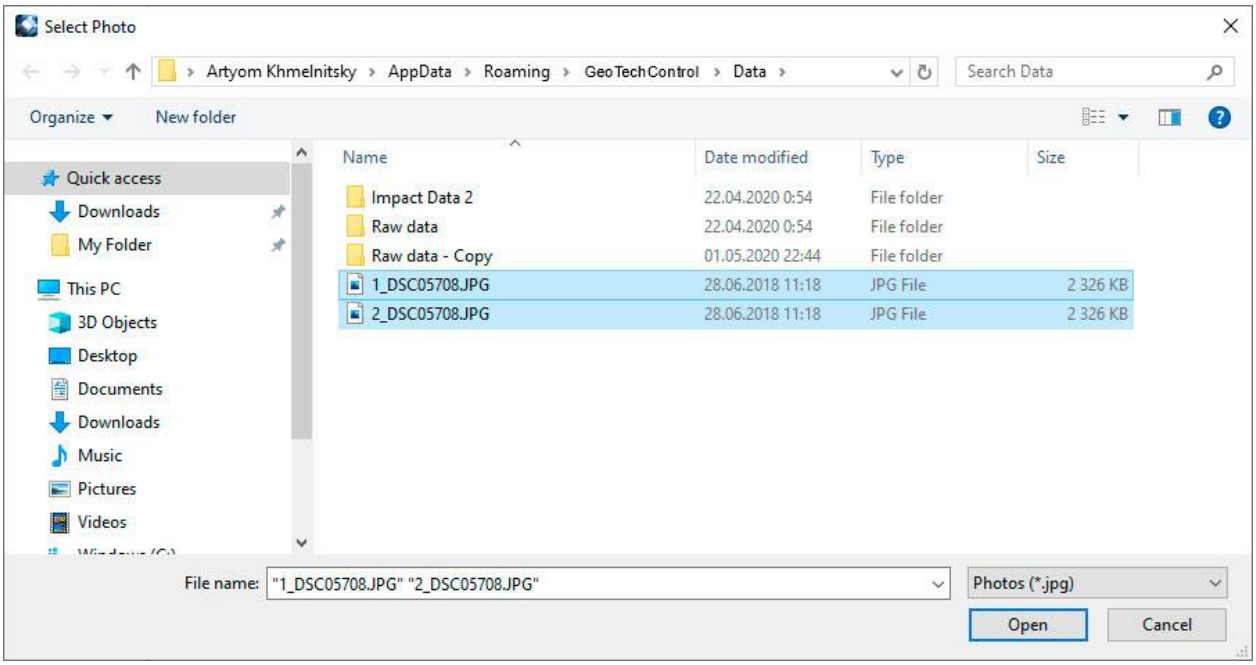

Fig. 20. Photo selection dialog box

In order to see the files added to the project, expand the folder that was selected before clicking "Add photos" (Fig. 21).

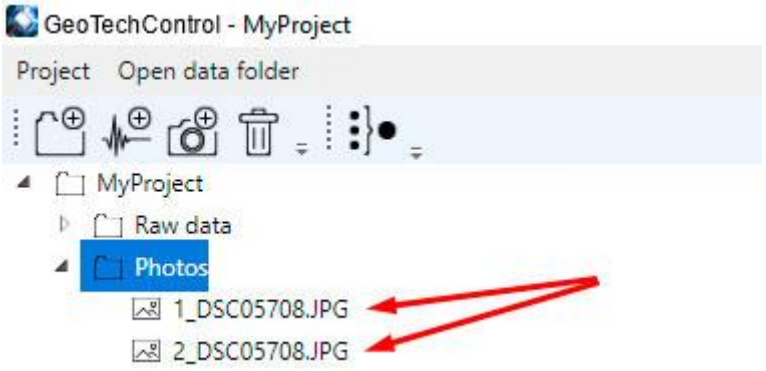

Fig. 21. New files added to the project tree

When uploading photos to the project, it is possible to automatically link them to piles.
In order to automatically link photos to piles, the following conditions must be met:

1. The required piles must be initialized in the project (see section 2.7). The piles in the project must be initialized for correct generation of the graphical report and binding of the measurement results to the surveyed piles.

2. The name of files with photos should start with the name of the pile that is in the project. After the name of the pile there should be a symbol "_" in the file name. For example, there is a pile in the project with the name "1". If the file name with the photo is "1_DSC05708.JPG", then the photo will be automatically linked to the pile "1" (Fig. 22)

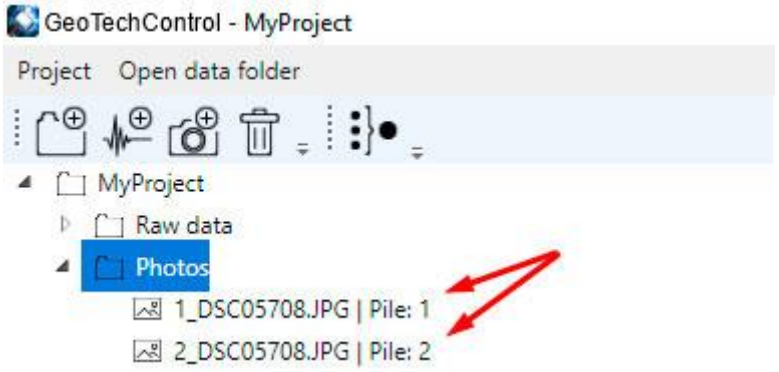

Fig. 22. Photos linked to piles

2.5. Viewing files

To view the seismic gather file, left-click on it in the project tree (Fig. 23).

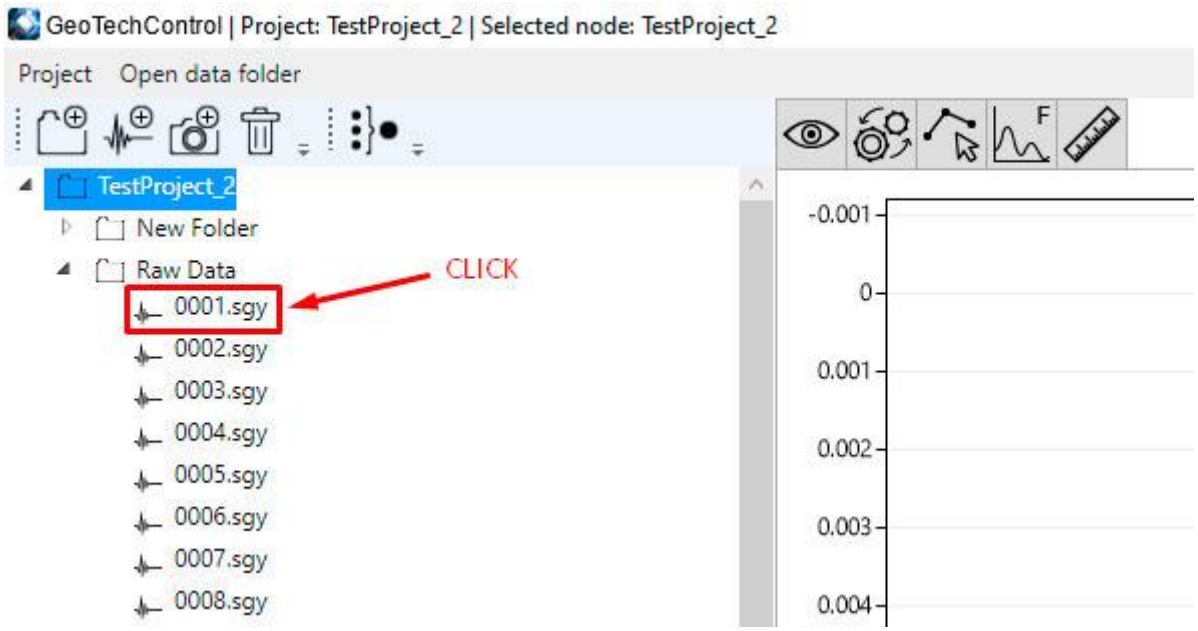

Fig. 23. Selecting a seismic gather file for subsequent viewing

As a result, the image of the selected seismic gather will appear in the signal area (Fig. 24).

Fig. 24. Viewing seismic gather file

To view the photo, double-click on it in the project tree (Fig. 25).

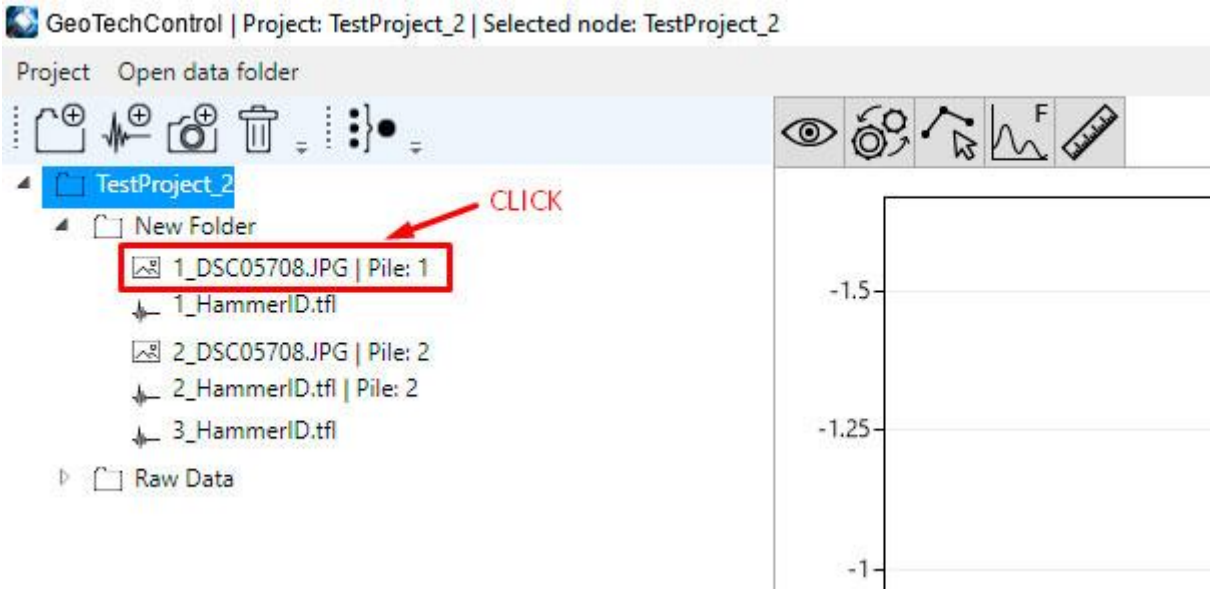

Fig. 25. Selecting a photo to view

As a result, the selected photo appears in the signal area (Fig. 26).

Fig. 26. Viewing a photo

2.6. Setting display of seismic gathers

The program allows to flexibly adjust the parameters of seismic gather display. To call the menu for setting the display parameters, display the gather on the screen and press the button "View configuration" of the toolbar in the main window of the program (Fig. 27).

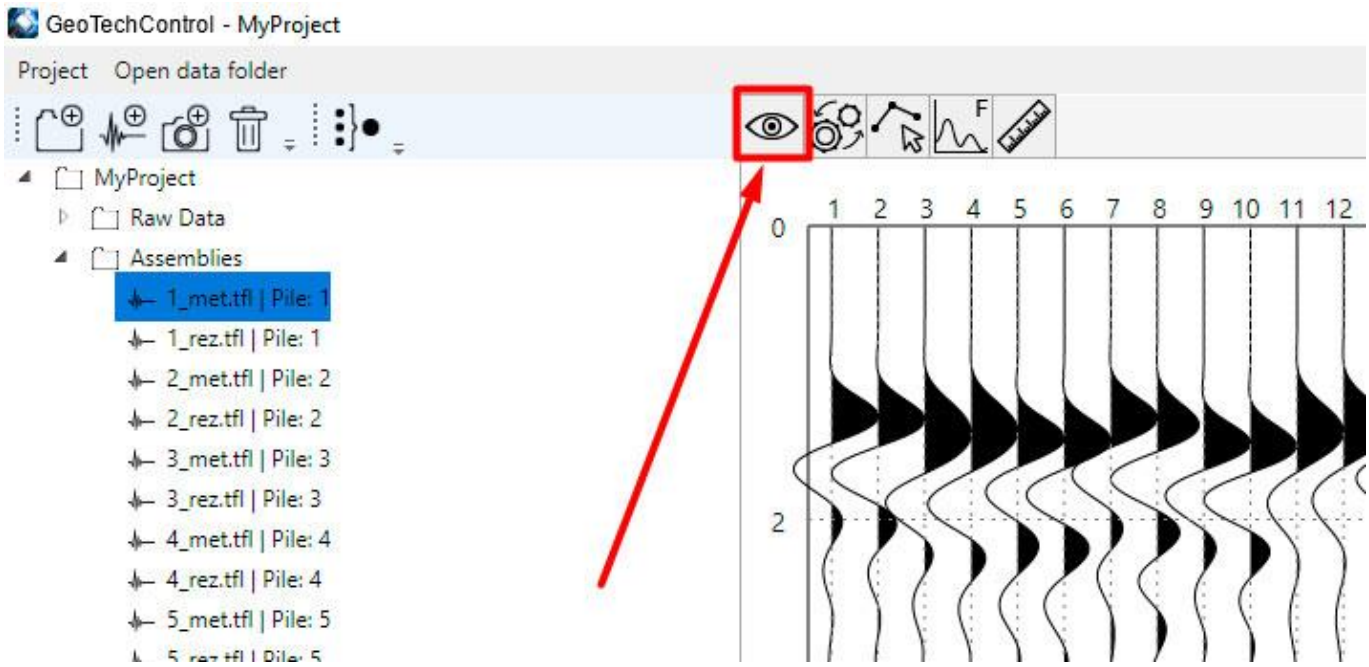

Fig. 27. Opening the menu for setting seismic gather display parameters

After clicking on the "View configuration" button a menu of display settings will open (Fig. 28).

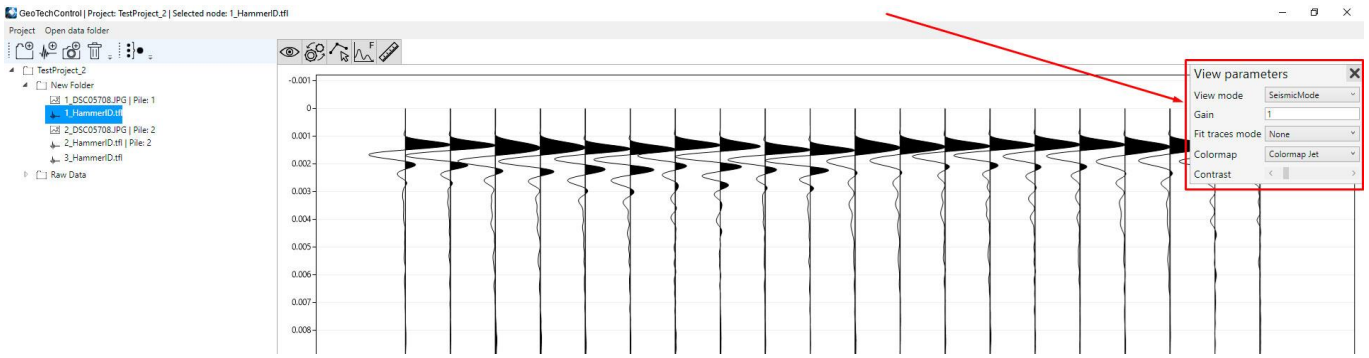

Fig. 28. Menu for setting seismic gather display parameters

You can select one of the five options for displaying the seismic gather (Fig. 29).

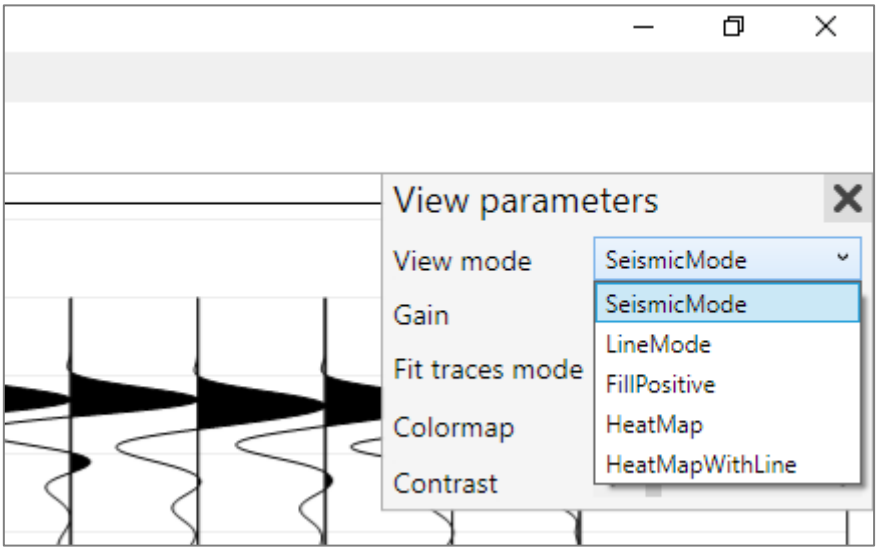

Fig. 29. Selecting the seismic gather display mode

Fig. 30 shows examples of what a seismic gather looks like for each of the five options:

А – Seismic mode.
Б – Fill Positive.
В – Single line.
Г – Heat map.
Д – Heat map + Simple Line

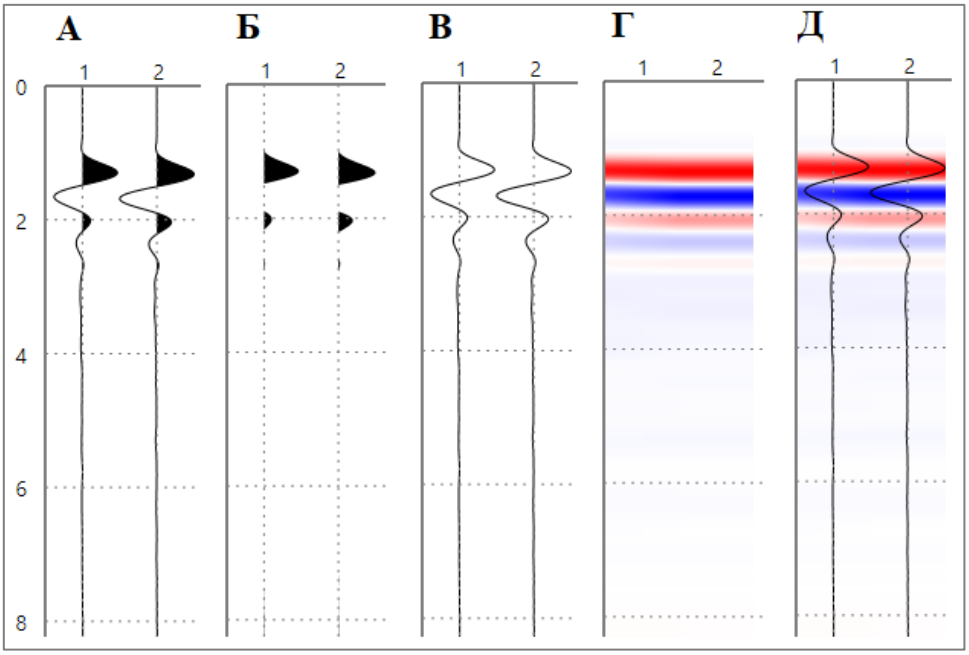

Fig. 30. Examples of seismic gather display in different modes

In addition to selecting the seismic gather display mode, you can control the following parameters:

1. Gain. Amplification when displaying the signal.
2. Fit traces mode. Additional modes of traces display (Fig. 31). None - traces are displayed "as is". Clip traces - traces are clipped to neighboring traces. Fit traces - the trace amplitude is aligned to the width of the area occupied by the trace.
3. Contrast. Changing the image contrast in HeatMap and HeatMapWithLine modes.

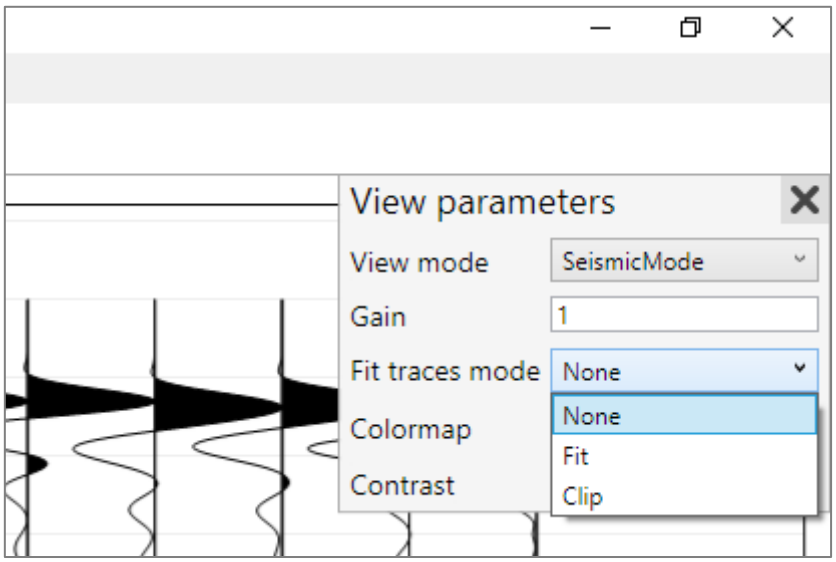

Fig. 31. Selecting trace alignment mode

To change the horizontal sweep of the graph, use the mouse wheel with the Ctrl key pressed.

To change the vertical sweep of the graph, use the mouse wheel with the Shift key pressed.

To change the signal location on the graph, drag the graph with the left mouse key pressed.

Press the middle mouse button to switch to the default settings.

The selected options are automatically applied to the opened files.

2.7. List of piles in a project

To build graphical reports and bind the seismic gathers interpretation results to them, it is necessary to initialize the piles in the project. The list of piles in the project is edited through the menu "Project - Project Settings" (Fig. 32).

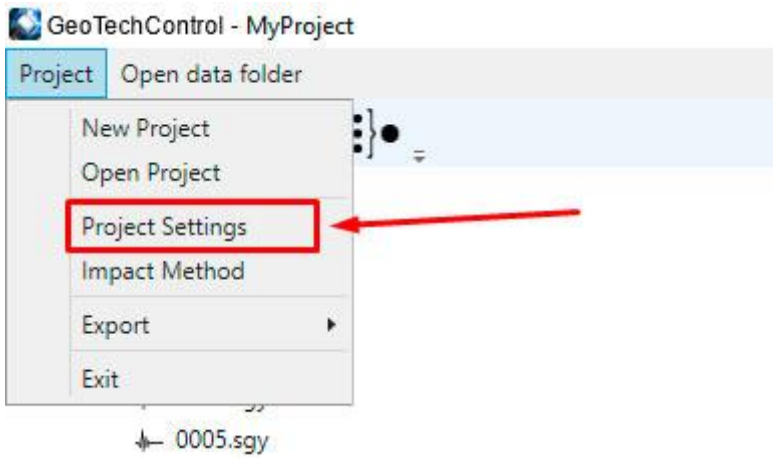

Fig. 32. Selecting "Project Settings" menu to edit the list of piles

By default, the list of piles in the project is empty.
To add a new pile click the "+" button (Fig. 33).

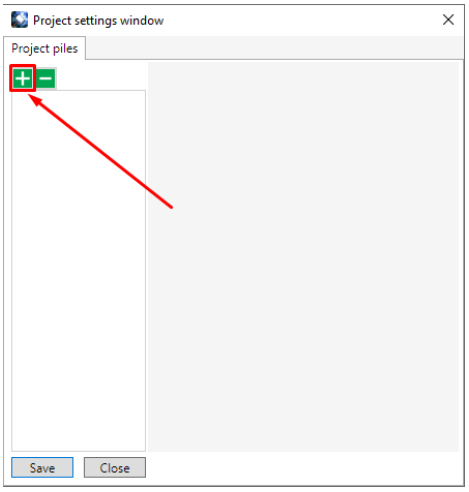

Fig. 33. Button to add a new pile in the "Project settings window"

In the appearing window, enter the name of the new pile and click "OK" (Fig. 34).

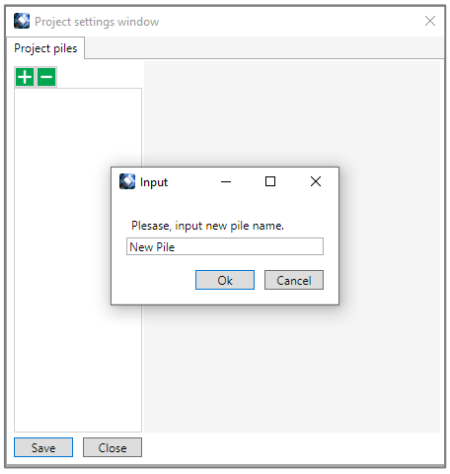

Fig. 34. Entering the name of a new pile

A new pile will appear in the list of piles (Fig. 35).

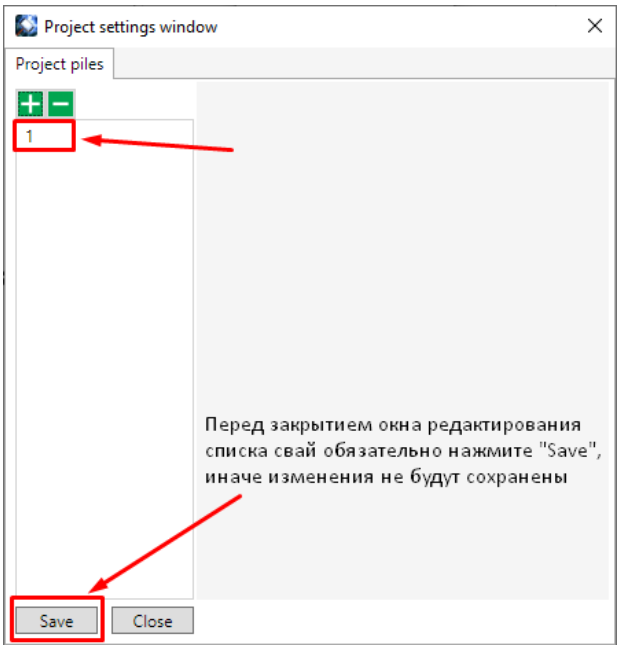

Fig. 35. Updated list of piles in the project

3 WORKING WITH FILES

3.1. Combining files into supergathers

When working with a dual-channel seismograph IDS-1, for each pile a set of files with two traces is recorded. The number of files is determined by the number of recordings repetitions. Before further data processing and interpretation, it is recommended to combine individual realizations into supergathers.

Before assembling seismic gathers it is necessary to add the raw data to the project (Fig. 36).

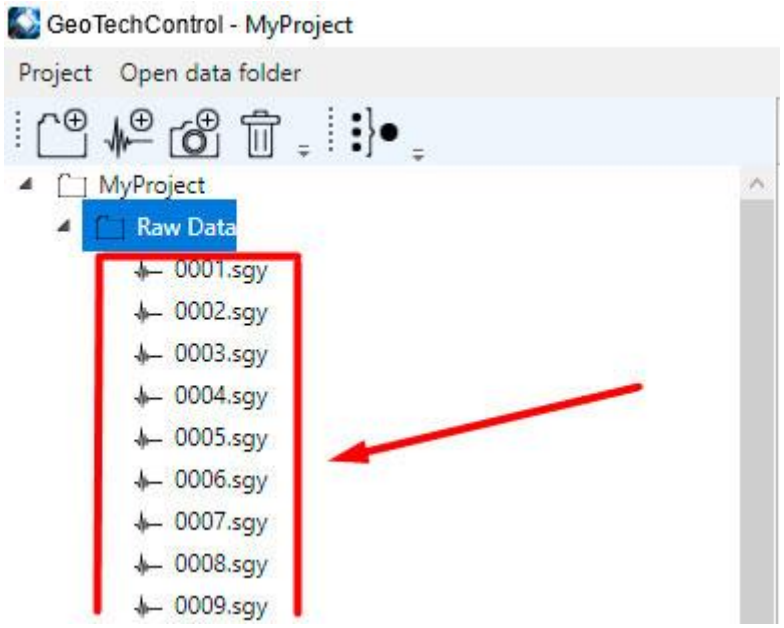

Fig. 36. Raw data, added to a project

In order to open the supergather window it is necessary to select the folder containing the raw data in the project tree and click the corresponding button (Fig. 37).

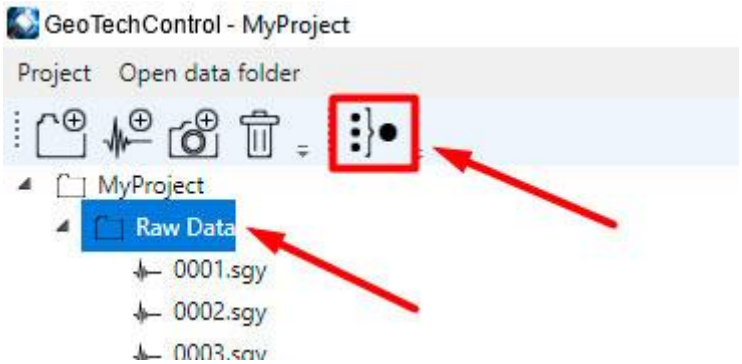

Fig. 37. Opening of the supergather window

After pressing the button to open the supergather window it will open (Fig. 38).

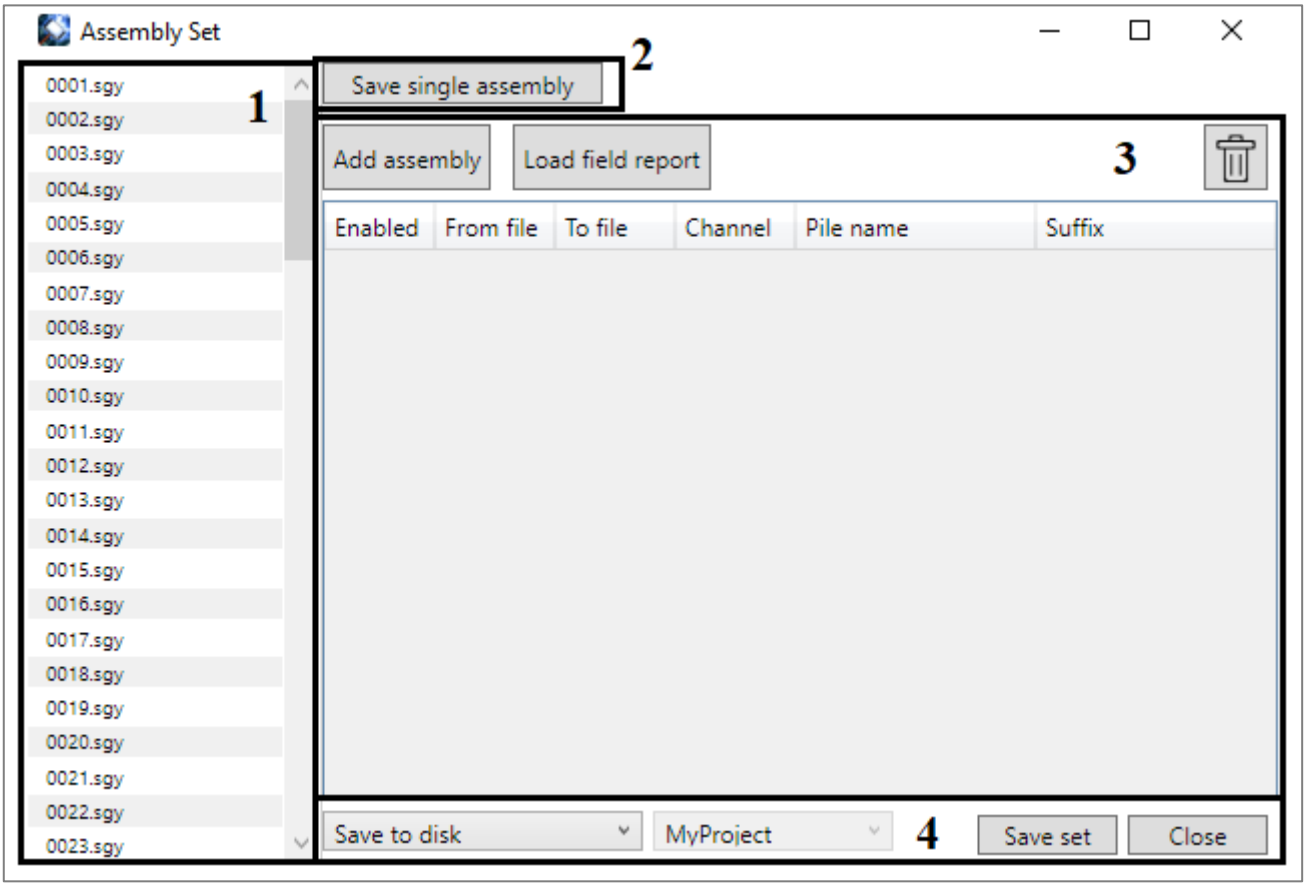

Fig. 38. Окно сборки сейсмограмм

The areas of supergather window are marked in Fig. 38: 1 – list of files in the selected folder; 2 – button for manual single assembly; 3 – area for automatic assembly of gathers using the field log file or via filling in the table; 4 – file saving control.

3.1.1. Manual assembly of seismic gathers

For manual single assembly of seismic gather select in the list the files to be merged and press the button "Save single assembly" (Fig. 39. *Единичная сборка сейсмограммы*)

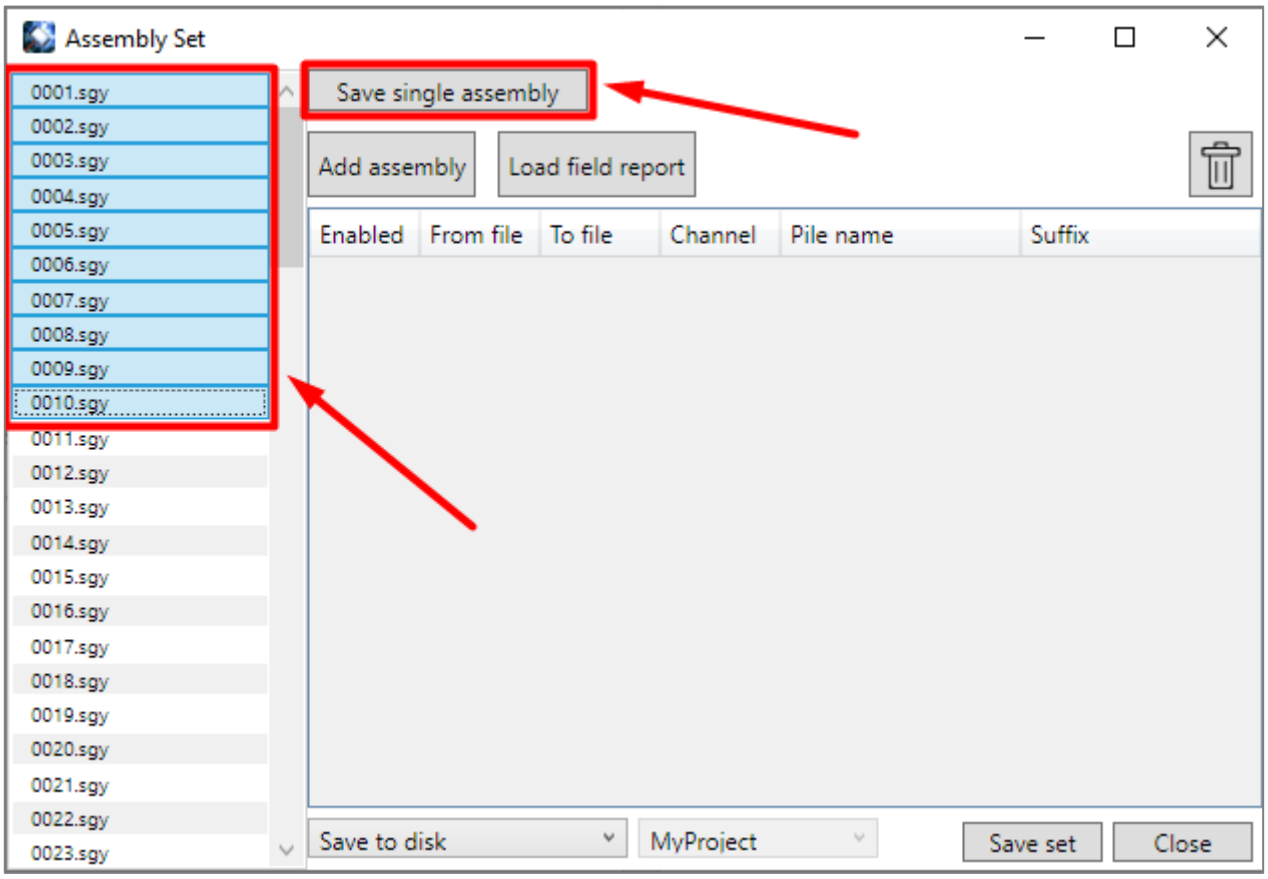

Fig. 39. Единичная сборка сейсмограммы

After clicking the "Save single assembly" button, the file-save dialog appears. Enter a file name (without an extension) and click "Save".

As a result, a file with a supergather will appear in the selected directory, in which the selected files will be combined. In case of manual assembly, the resulting file contains all the traces and channels recorded in the source files. The assembly files have the ".tfl" extension.

To view the assembly in the program, add the assembly file to the project (see section 2.4.2). If the described operations are performed successfully, you will be able to see the created assembly in the program (Fig. 40). In the above example, the source files are the realizations recorded by the two-channel seismograph IDS-1. As a result of assembling 10 two-channel recordings, we got an assembly consisting of 20 seismic traces.

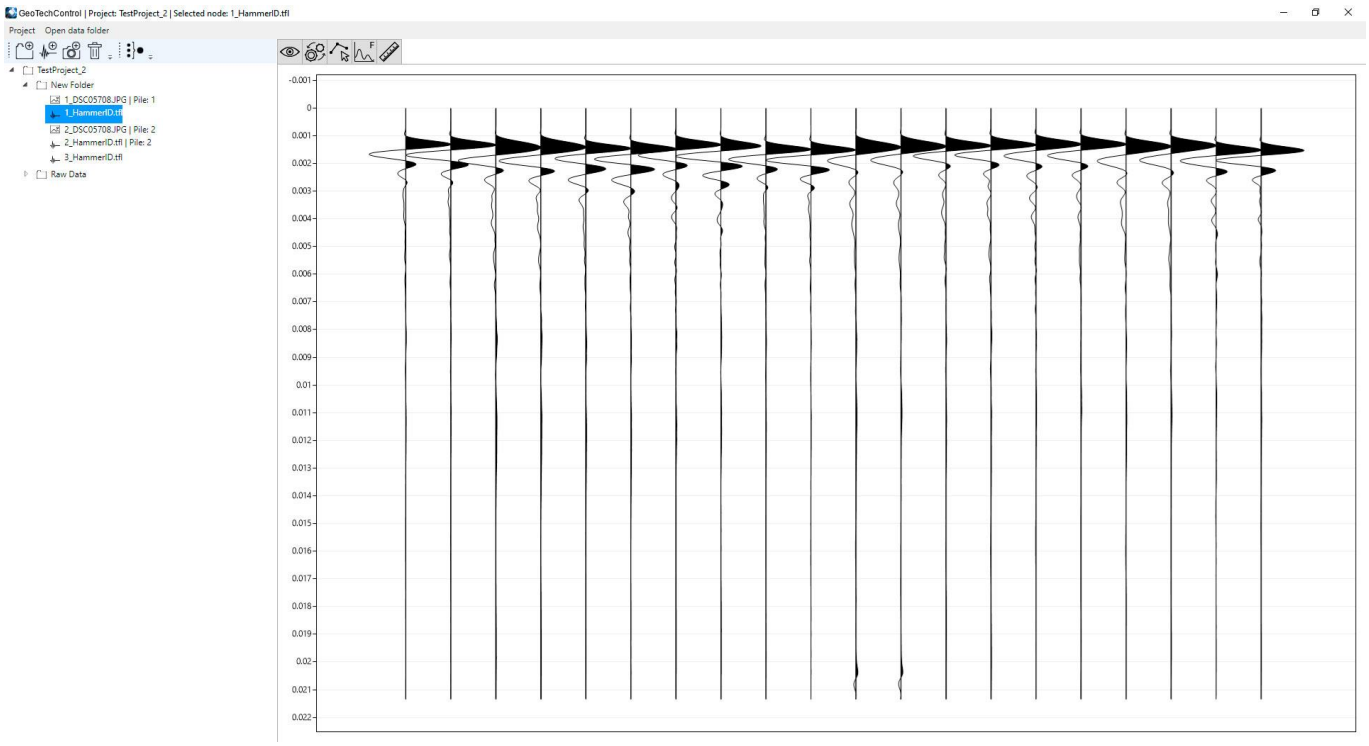

Fig. 40. Supergather

3.1.2. Automatic assembly of seismic gathers through the interface

In the program there is a functionality allowing to automatically collect many seismic gathers at once. For this purpose it is necessary to fill in the table specifying the order of assembly in the window of gather assembly. For this purpose it is necessary to fill in the table specifying the order of assembly in the supergather window.

Filling the assemblies table is possible in two ways:

1. Manually via the program interface
2. Automatically using a field log file

In order to be able to assemble files automatically, file names must be numbers without other characters. For example: 0001.sgy, 0002.sgy, …

To add an assembly to the table through the program interface, click the "Add assembly" button. After clicking the button, a new line will be added to the table of assemblies (Fig. 41).

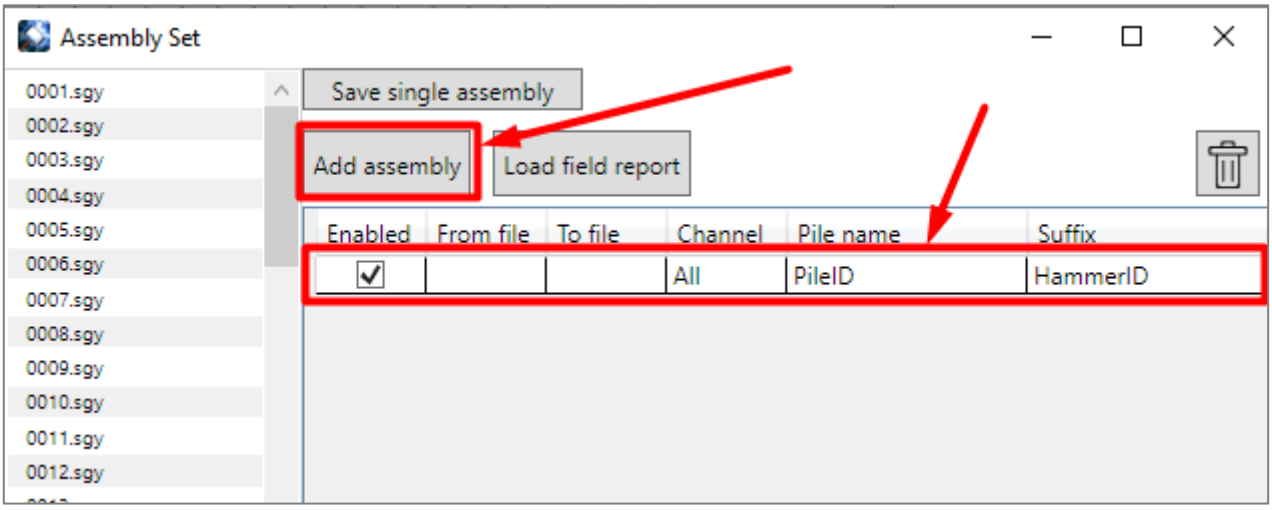

Fig. 41. Adding an assembly to the table

Each row in the table is one assembly.
A row has the following attributes:

1. Enabled. If this attribute is turned on for a row, then this row will be taken into account during assembly. If this attribute is disabled, then this assembly will not be created.
2. From file. The number of the file from which the assembly starts. A number, without an extension, is specified.
3. To file. The number of the file with which the assembly ends. A number, without an extension, is specified.
4. Channel. The channel number to be taken from the files and used in the assembly. Either a number denoting the channel number or an "All" raw denoting the use of all channels from the files is specified. From each file in the selected range from "from file" to "to file" it will be taken the channel(s) specified in the "Channel" attribute.
5. Pile name. Identifier of the pile to which this assembly belongs.
6. Suffix. Assembly identifier is needed to distinguish assemblies by some attribute. For example, if a series of records with different sources were taken on the same pile, then Suffix can be the source identifier.

Add to the table the required number of assemblies and fill in the parameters of the assembly. For an example of the completed table, see (Fig. 42).

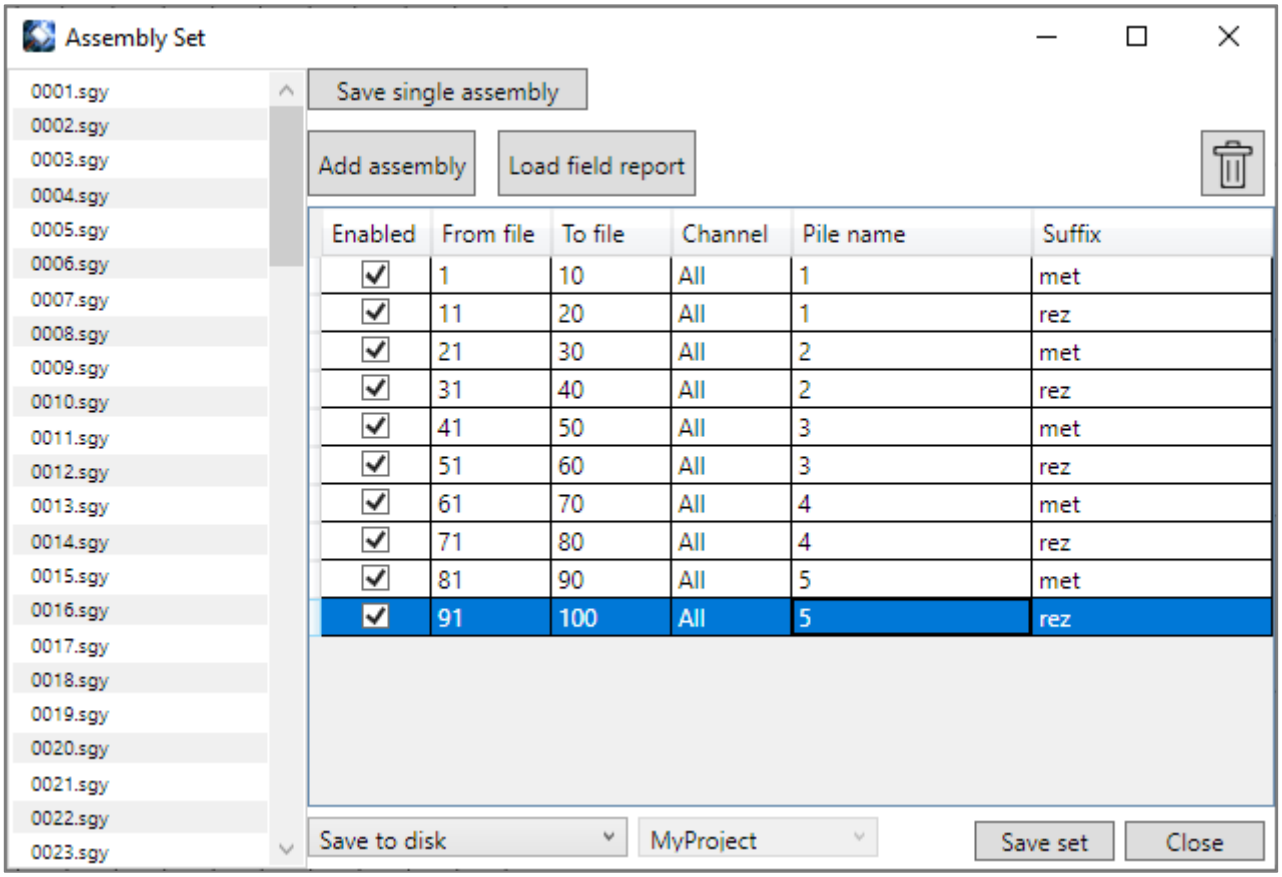

Fig. 42. Example of a filled assembly table

To delete a row from the table, select the needed row and click the "delete" button in the upper right corner of the window above the table.

After completing the table, you should run the procedure of forming and saving the assemblies. Select where you want to save the assemblies.

Two options are available for saving assemblies:

1. to the computer's hard drive,
2. to a folder in the project

Saving assemblies to your computer's hard drive

In order to save to the hard disk, the corresponding option should be selected in the list of save locations (Fig. 43).

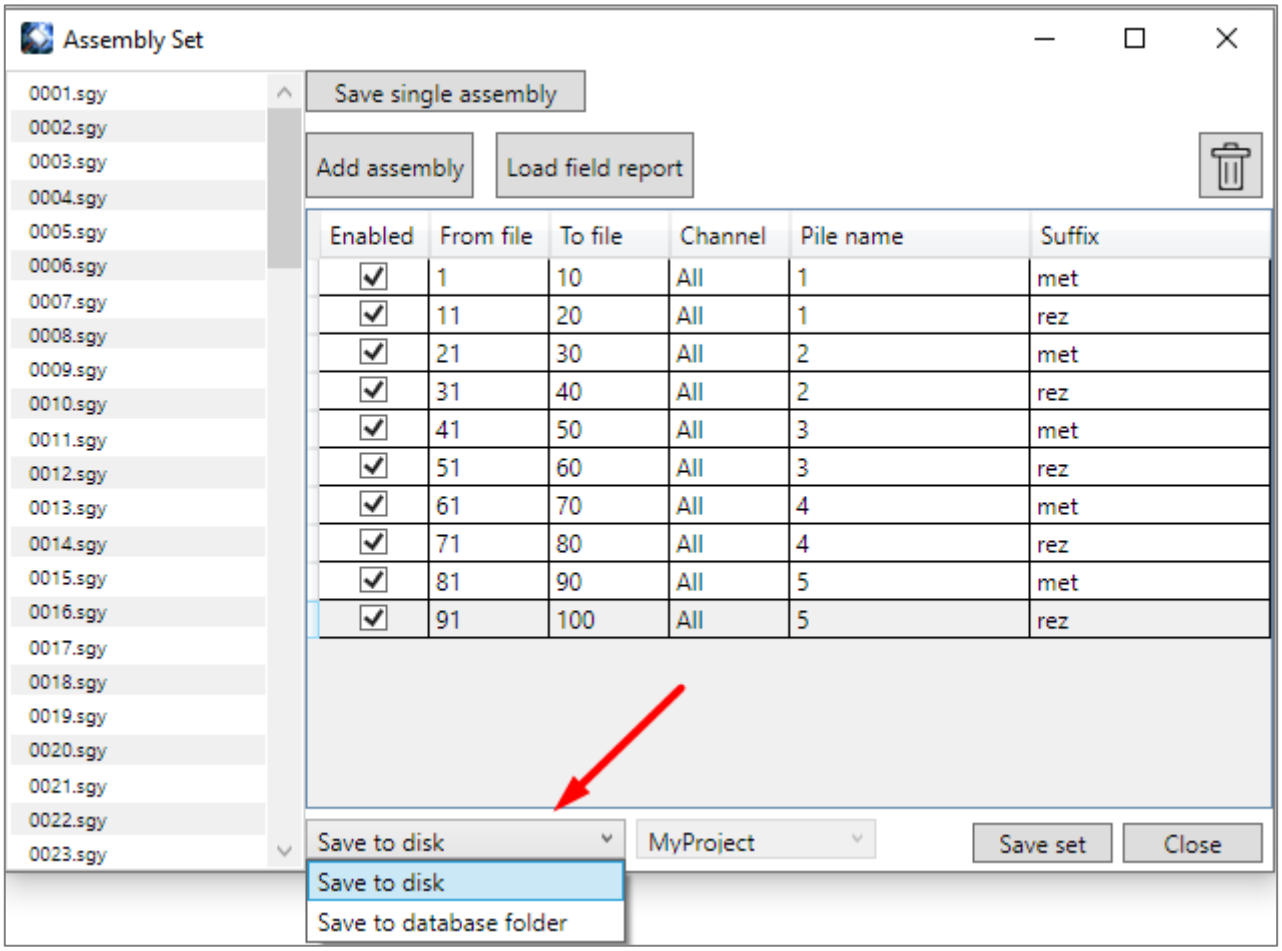

Fig. 43. Choosing where to save the assemblies

After the save location is selected, click the “Save set” button. The dialog box for selecting a folder for saving the supergathers will open. Select the desired folder and click "Select folder". The files corresponding to the filled table will appear in the selected folder (Fig. 44).

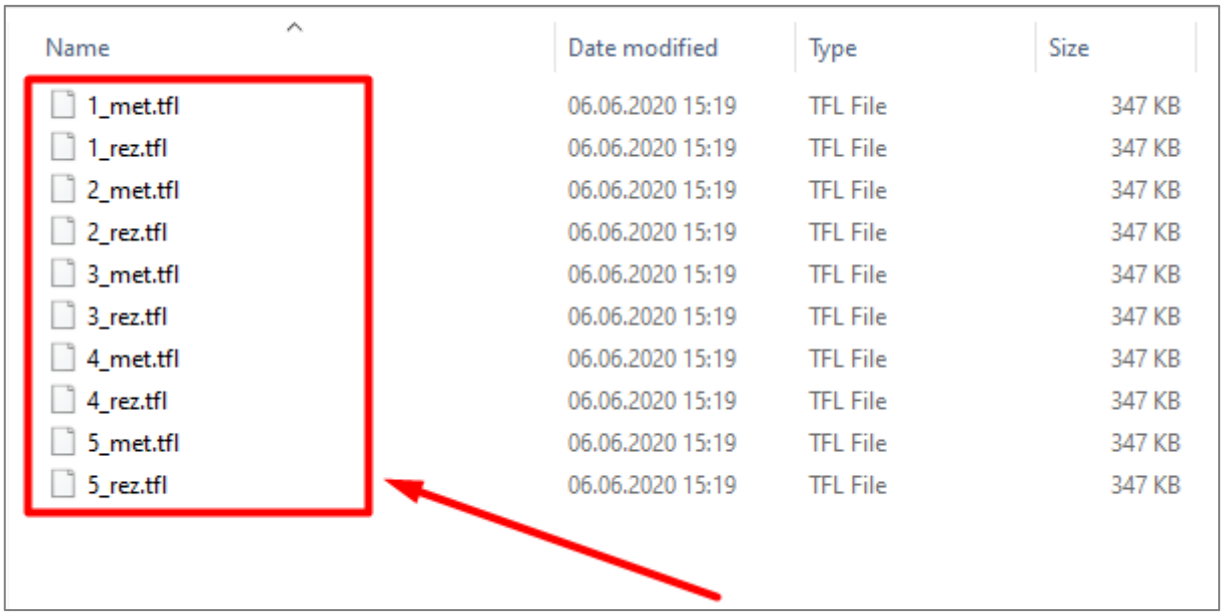

Fig. 44. Supergathers recorded in the selected folder

The name of each supergather is formed according to the attributes "Pile name" and "Suffix" set in the table of assemblies.
For example, "Pile name" is "1", and "Suffix" is "met". The resulting name of such an assembly would be "1_met.tfl".

Saving assemblies to a folder in the project

To save the assemblies to a folder in the project, select "Save to database folder" in the Save location list. In the second list select the folder in the project to save the assemblies to (Fig. 45). The required folder, where the assemblies will be added, must be previously created in the project tree.

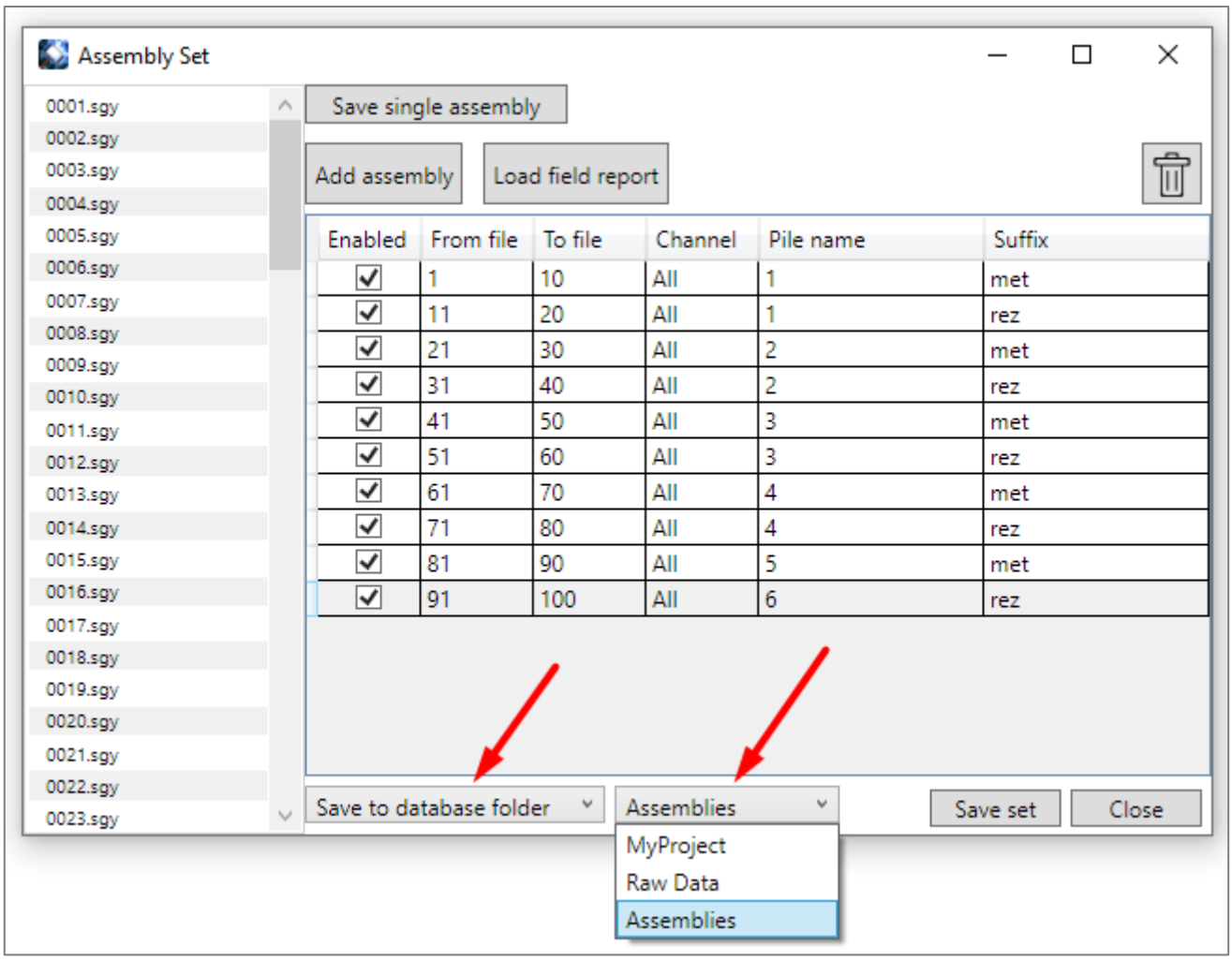

Fig. 45. Saving assemblies to a folder in the project

Click "Save set", and after the supergathers are successfully completed, they will be added to the selected project folder (Fig. 46).

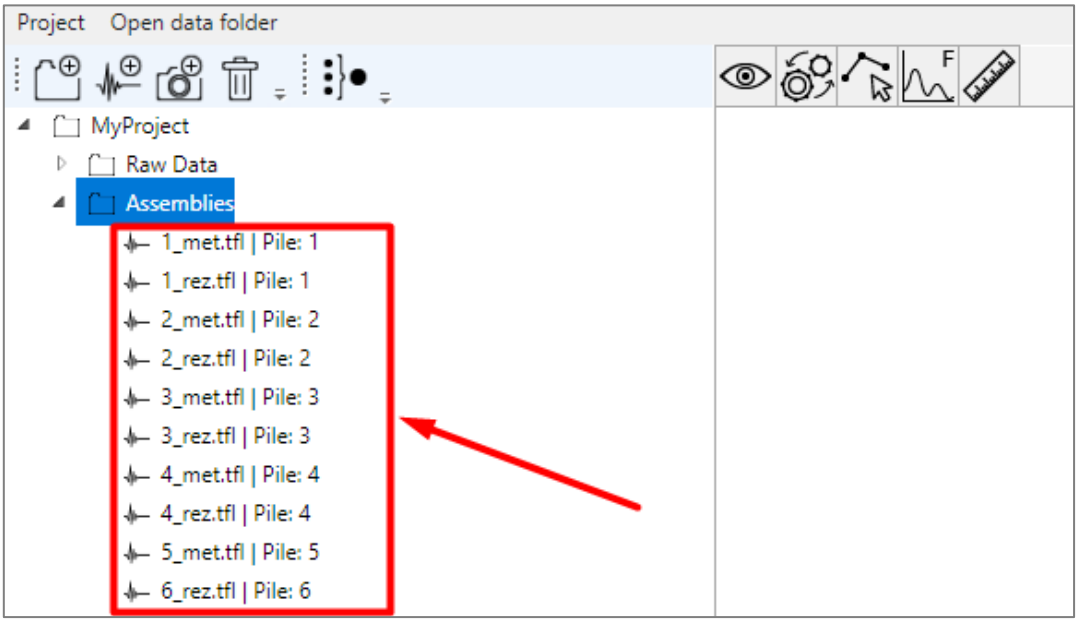

Fig. 46. Supergathers added to a folder in the project

In case of saving seismic gathers during assembly directly to the folder in the project, it is possible to automatically link the assemblies with piles. For automatic binding, the assembly attribute "Pile name" should be the same as one of the pile names added to the project at the time of assembly. For example, the project already has piles named "1", "2", "3", "4" and "5". If you do all the steps in the same way as mentioned above, the collected gathers will be automatically bound to the piles. In the project tree, a reference to the linked pile is added to the file name (Fig. 46).

3.1.3. Automatic assembly of seismic gathers using a field log

Forming assemblies in the case of using the field log is similar to the assembly by filling the table of assemblies through the interface. The difference is that the table of assemblies is filled by loading into the program the field log, which contains all the necessary information for the assembly.

The field log file is a .csv text file.
Below is an example of a field log file

From;To;Channel;Pile;Suffix;
1;10;all;pile-1;met;
11;20;all;pile-1;rez;
21;30;all;pile-2;met;
31;40;all;pile-2;rez;
41;50;all;pile-3;met;
51;60;all;pile-3;rez;
61;70;all;pile-4;met;
71;80;all;pile-4;rez;
81;90;all;pile-5;met;
91;100;all;pile-5;rez;
91;100;all;pile-5;rez;

The file must have the first line with headers.
Each line in the file defines one supergather.
For example: 11;20;all;pile-1;rez;
In this example, all channels in realizations 11 through 20 will be assembled into one seismic gather, and the name of the assembly file will be "pile-1_rez.tfl".

To load the field log, click "Load field report" and select the field log file in the dialog box that appears (Fig. *47*).

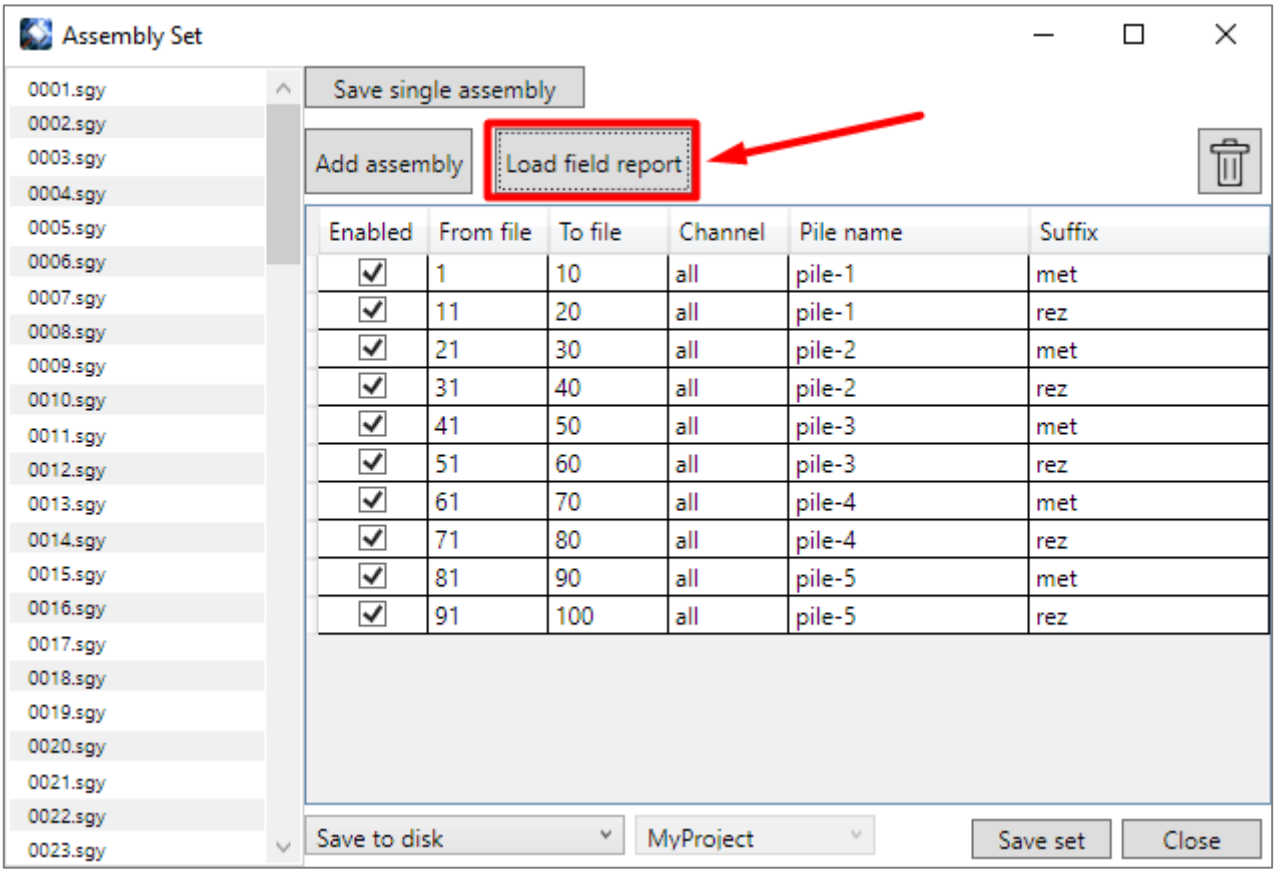

Fig. 47. Loading the field log file and the assemblies table filled with it

Then the files with assemblies are saved in the same way as described above.

3.2. Signal processing

The program has a set of procedures for digital signal processing.
To open the configuration window of digital signal processing, select the seismic gather in the project tree by double-clicking the left mouse button and click the corresponding button on the toolbar (**Fig.** 48).

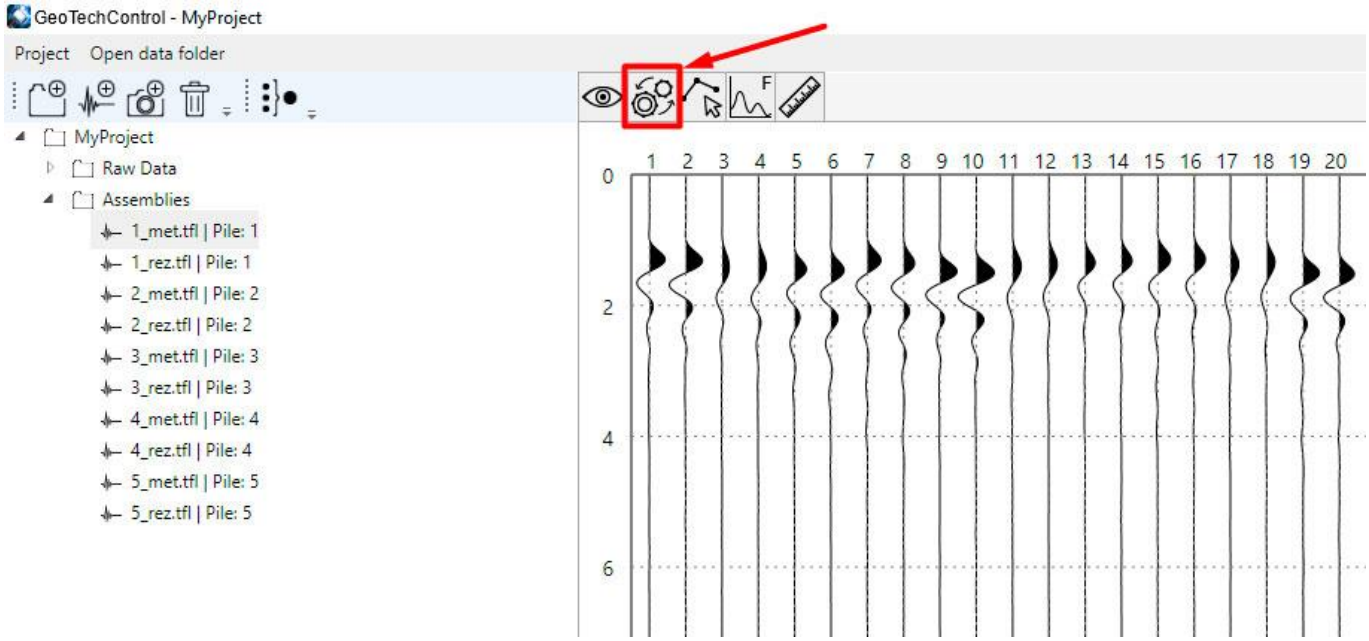

Fig. 48. Opening the digital signal processing configuration window

The filter settings menu will open. Select the desired filter in the filter list and add it to the processing thread by pressing "+" (Fig. 49).

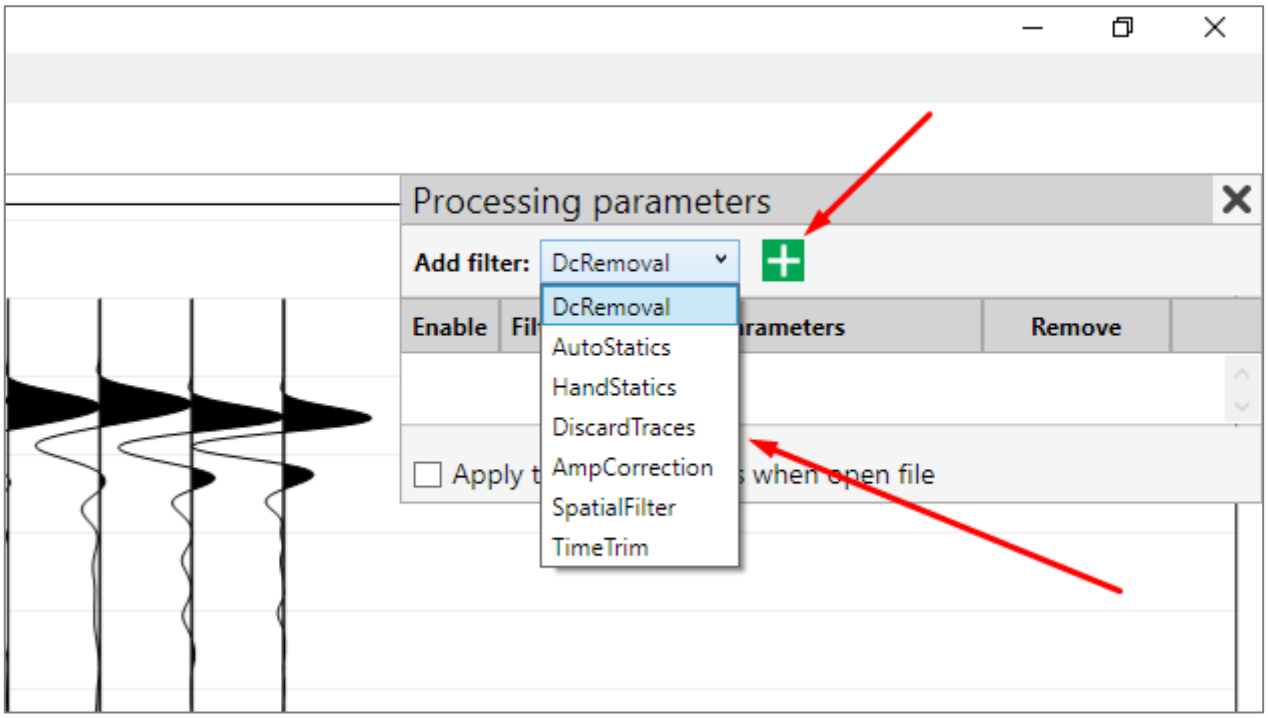

Fig. 49. Adding filter to the processing thread

Add filters to the processing thread until it is fully formed (Fig. 50).

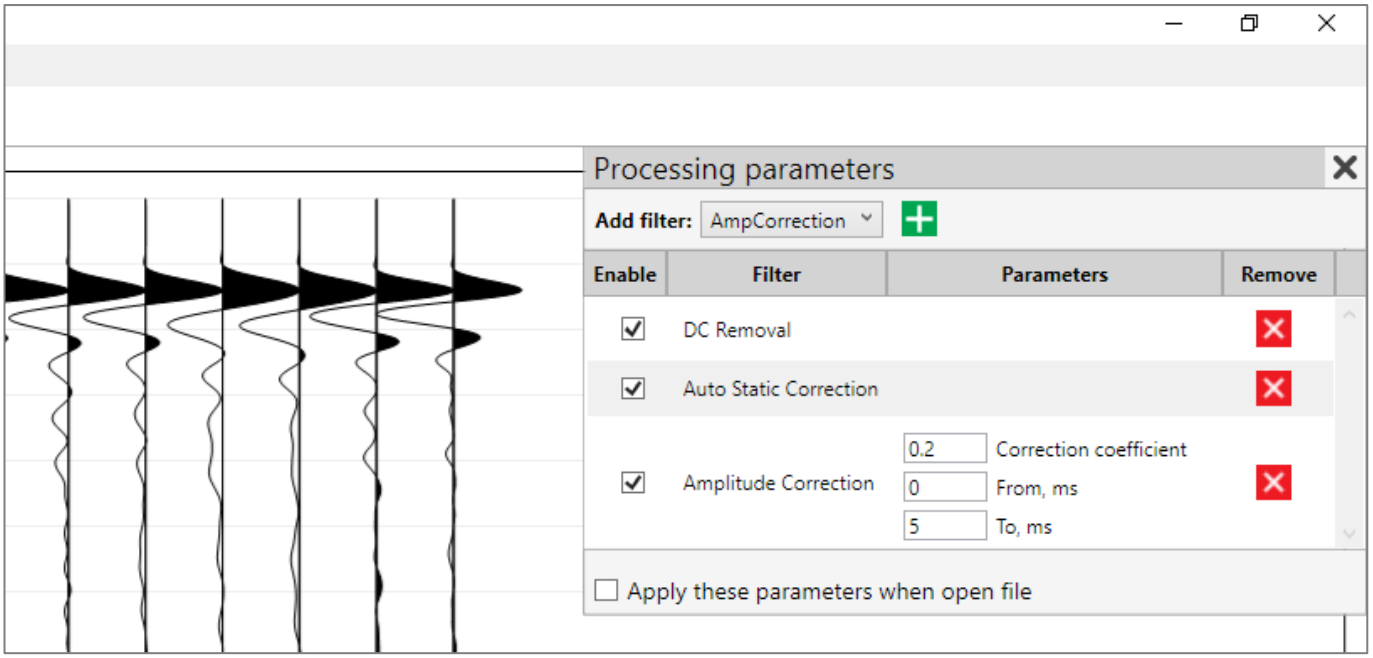

Fig. 50. Example of a generated processing thread

The processing thread is automatically applied to the file when any change is made.

To disable a filter from the processing thread, in the "Enable" column, uncheck the corresponding filter.

To remove a filter from the processing thread, in the "Remove" column click "X" for the corresponding filter.

3.2.1. DC removal

To remove the DC component from the signal, use the "DC Removal" filter (Fig. 51). This filter has no settings.

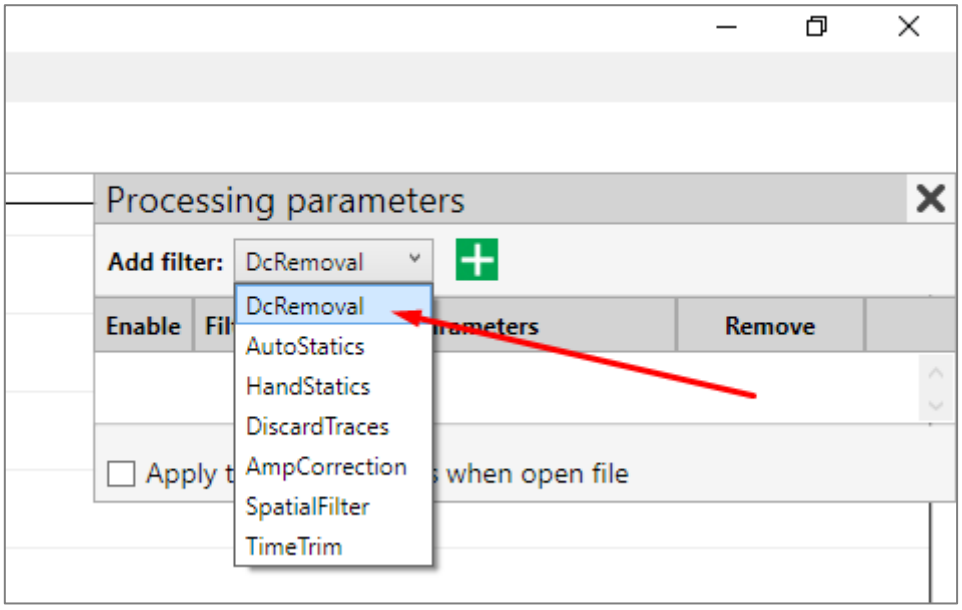

Fig. 51. Selecting a DC removal filter

3.2.2. Automatic static correction

Use the "AutoStatics" filter (Fig. 52). to enter automatic static correction. This filter has no settings.

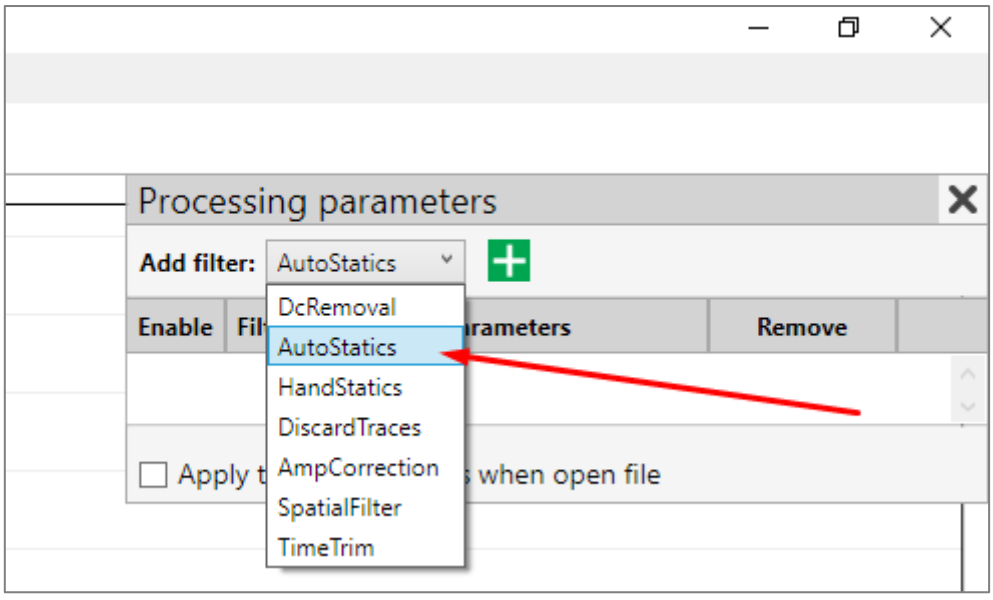

Fig. 52. Selecting manual static correction filter

An example automatic static correction procedure result is shown in Fig. 53.

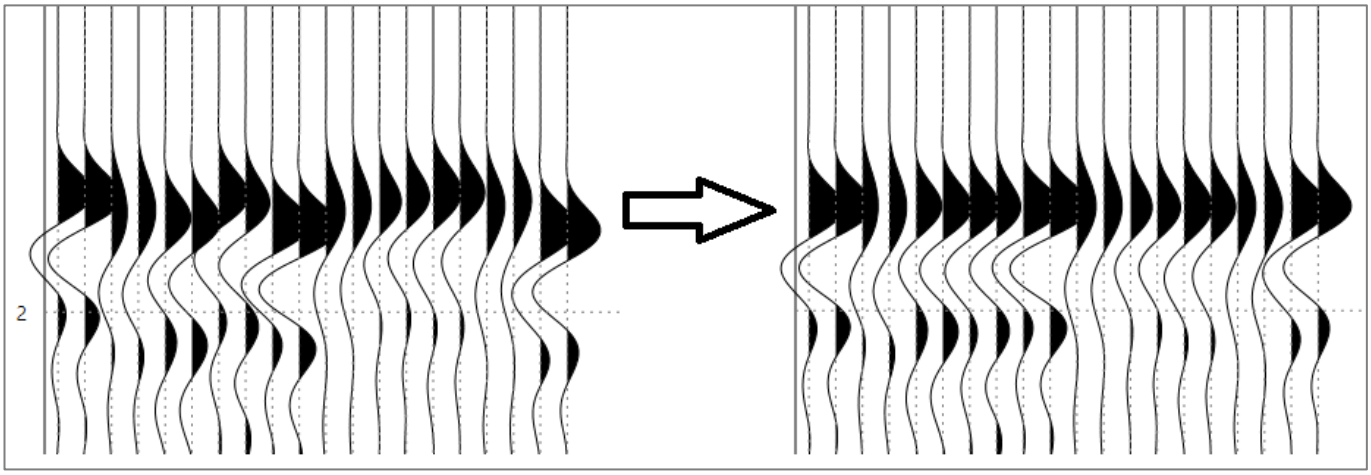

Fig. 53. Example of applying automatic static correction

3.2.3. Manual static correction

Для ввода автоматической статической поправки воспользуйтесь фильтром “Auto Static Correction” (Fig. 54).

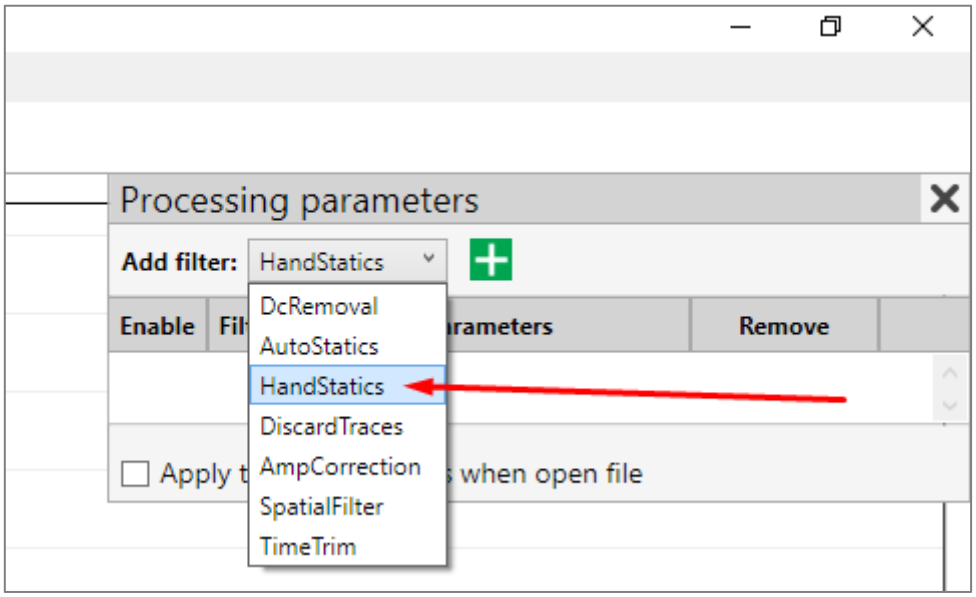

Fig. 54. Selecting manual static correction filter

Manual static correction input is available in two variants:

1. By picking.
2. By time value.

To enter a static picking correction, you need firstly to enter the picking for the file (see the section “First break picking and trace rejection and trace rejection”) and activate the "Use peak" filter parameter (Fig. 55).

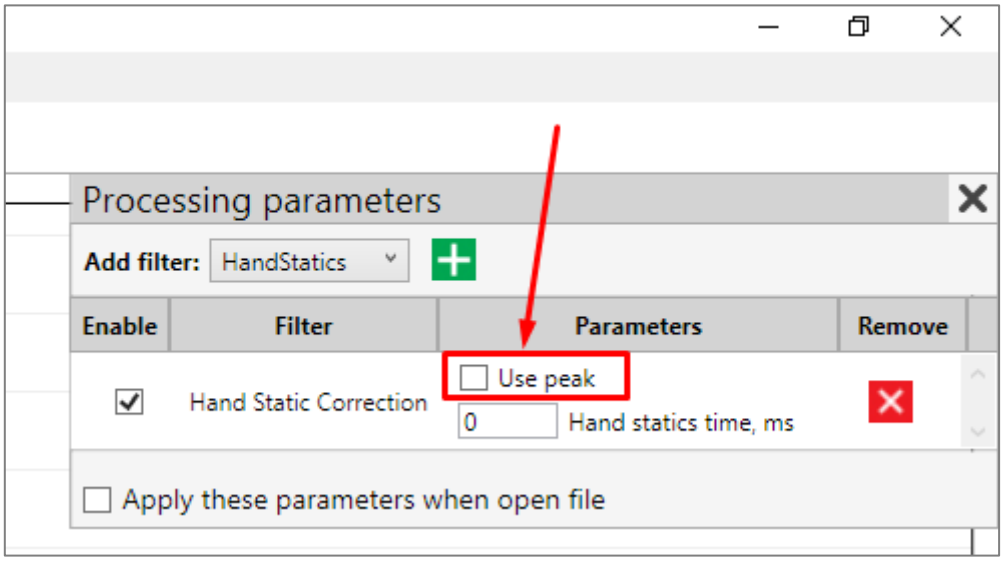

Fig. 55. Selecting the first break picking stacking correction input

An example of the result of entering the first break picking static correction is shown Fig. 56

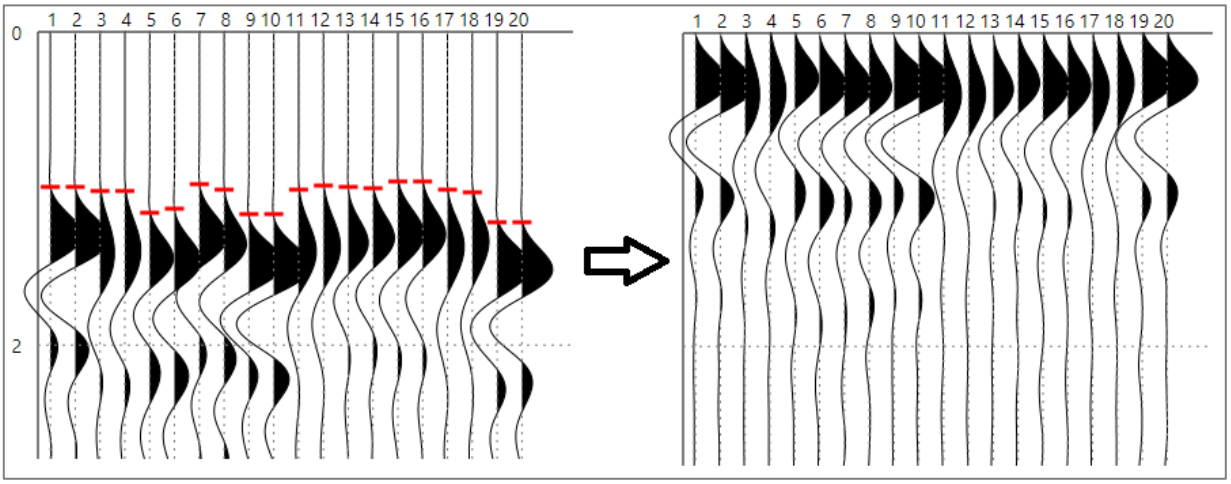

Fig. 56. Example of applying first brake picking static correction

To enter the static correction by time value, deactivate the "Use peak" parameter of the filter and add the time value to the "Hand statics time" window (Fig. 57).

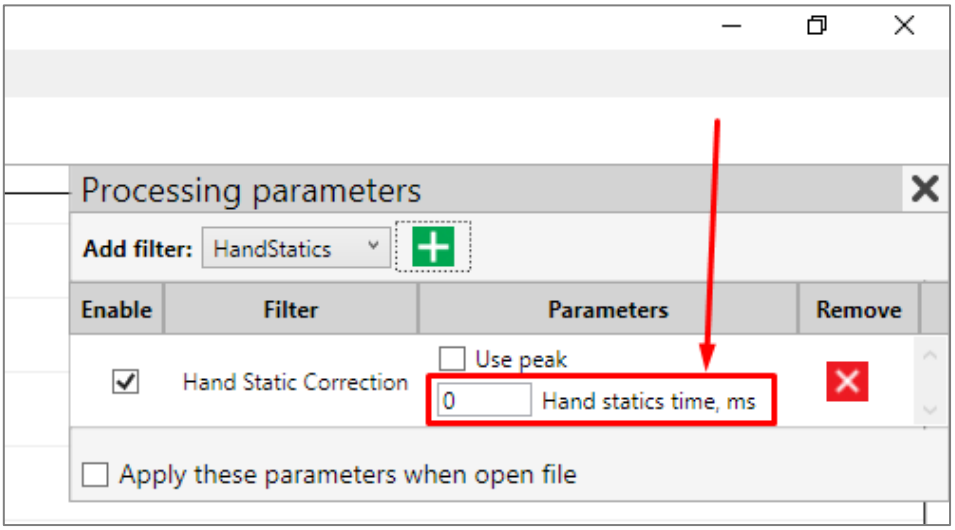

Fig. 57. Entering the time value for manual static correction

An example of the result of entering the static correction by time value is shown in Fig. 58.

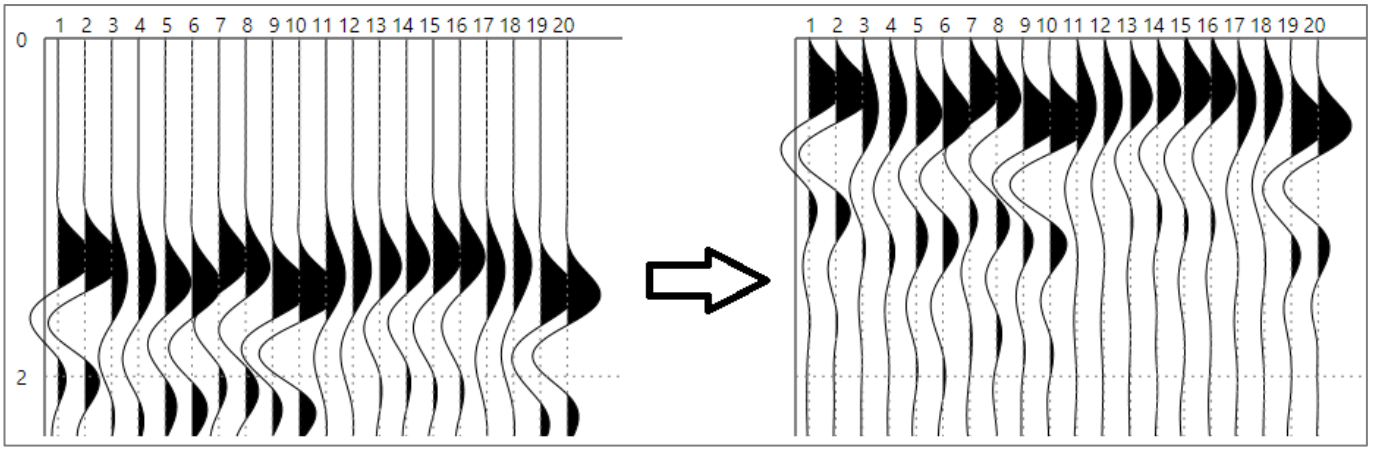

Fig. 58. Example of entering the static correction by time value

3.2.4. Trace rejection

The trace rejection procedure allows one to exclude some traces from processing.
In order to apply this procedure, you need first mark the defective traces (see section "First brake piking and and trace rejection").

To apply the trace rejection procedure, use the "Discard Traces" filter (Fig. 59).

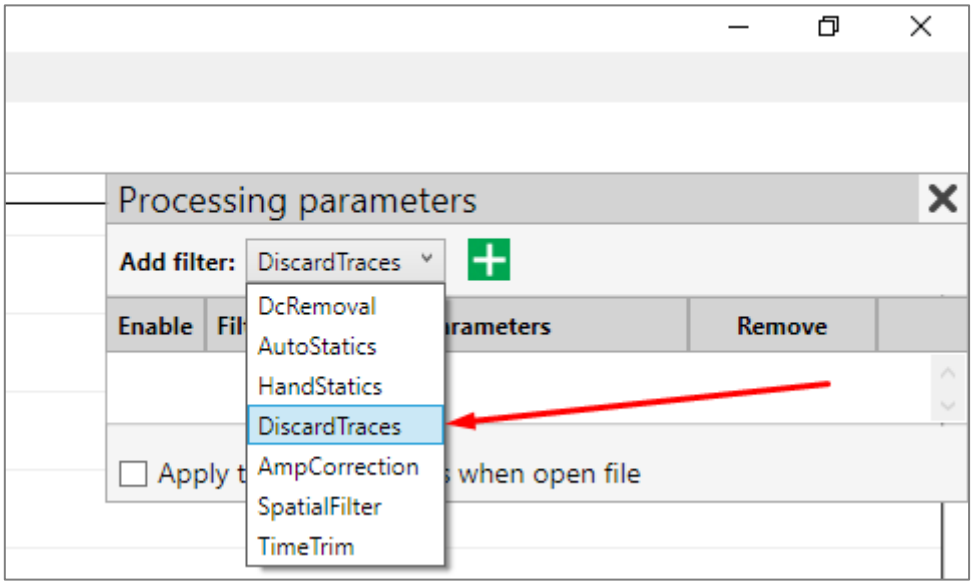

Fig. 59. Selecting the trace rejection filter

All filters standing in the signal processing thread after the trace rejection filtering will be applied to the trace set that lacks the marked traces. An example of applying the trace rejection procedure is shown in Fig. 60.

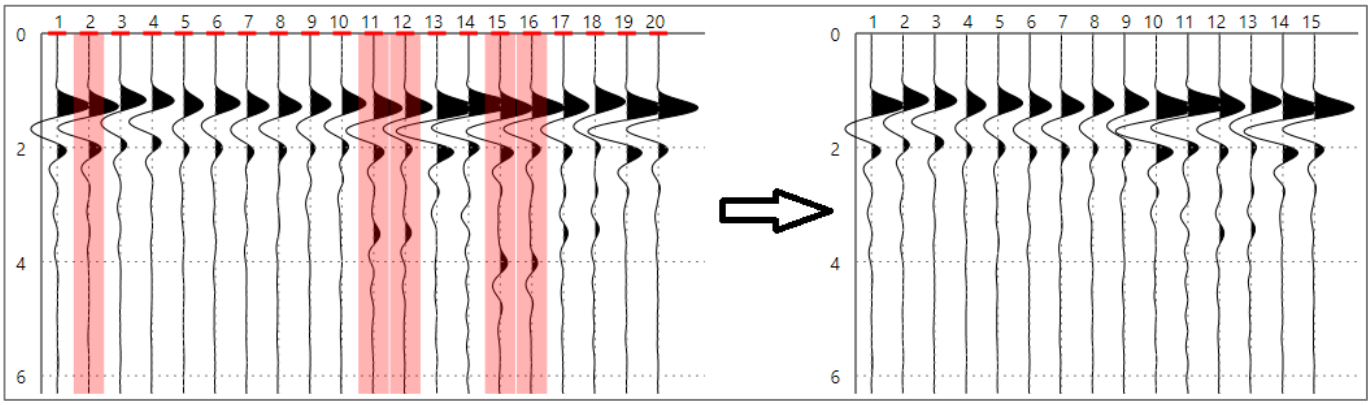

Fig. 60. Example of applying the trace rejection procedure

3.2.5. Amplitude correction

The amplitude correction procedure is used to compensate for signal attenuation.
To apply the amplitude correction procedure, use the "Amplitude Correction" filter (Fig. 61).

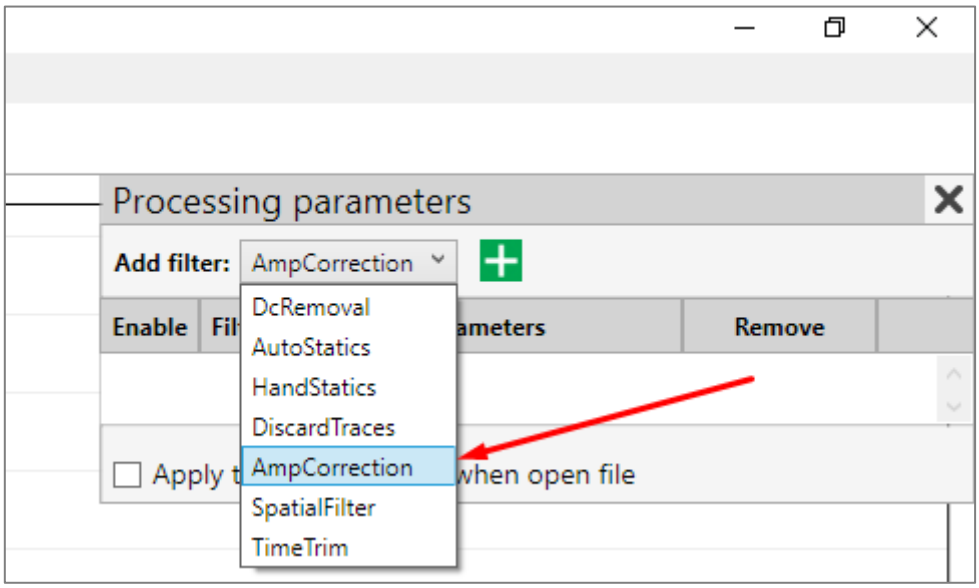

Fig. 61. Selecting the amplitude correction filter

The amplitude correction has two settings (Fig. 62):
1. Correction coefficient. This coefficient is the degree of the exponential gain profile that corrects the signal's amplitude.
2. Window for calculating the signal amplitude, which is applied when aligning traces. The values are set in milliseconds.

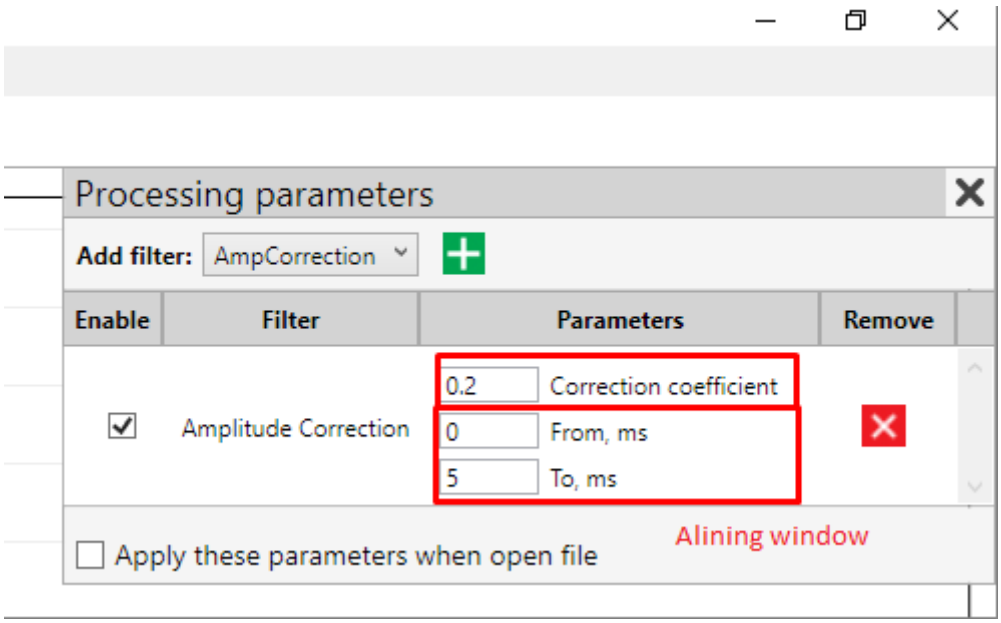

Fig. 62. Amplitude correction filter settings

When working with files recorded by seismograph IDS-1, it is recommended to use the correction coefficient from 0.1 to 0.4 at a 20 ms sweep.
For trace alignment, it is recommended to specify a window that includes the first break of the signal.
An example of applying the amplitude correction procedure is shown in Fig. 63.

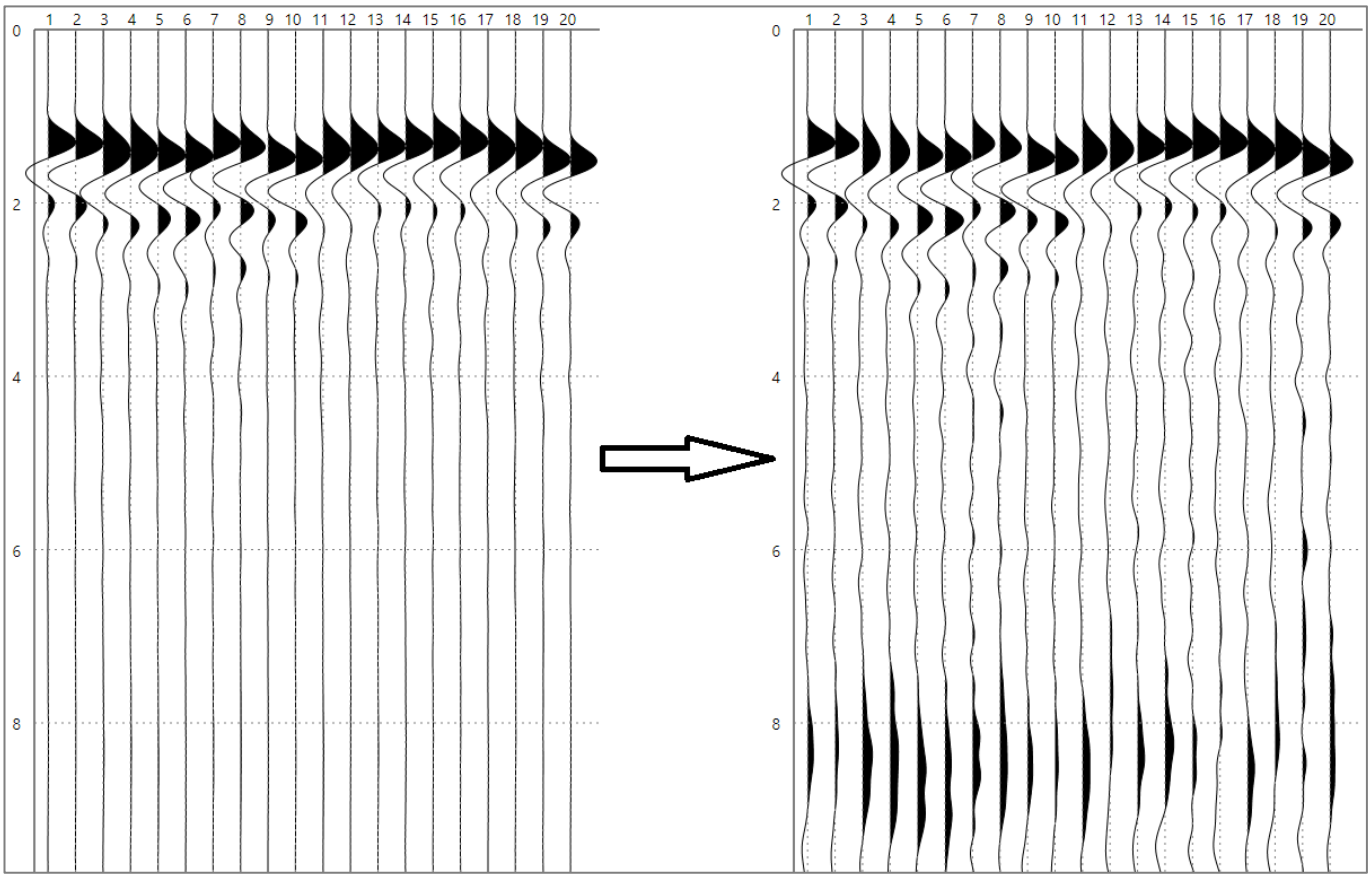

Fig. 63. Example of applying the amplitude correction procedure

3.2.6. 2D spatial filtering

To apply the smoothing procedure in the 2D sliding window, use the "Spatial Filter" procedure (Fig. 64).

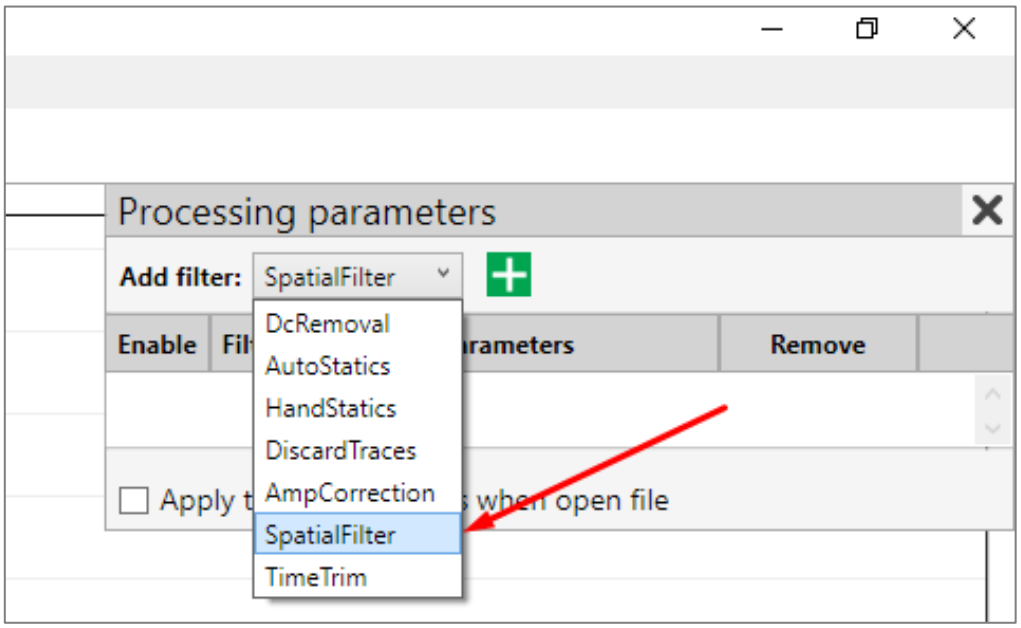

Fig. 64. Selecting the smoothing filter in the 2D sliding window

The smoothing filter in the 2D sliding window has two settings:

1. Number of samples.
2. Number of traces.

These two settings determine the height and width of the smoothing window.

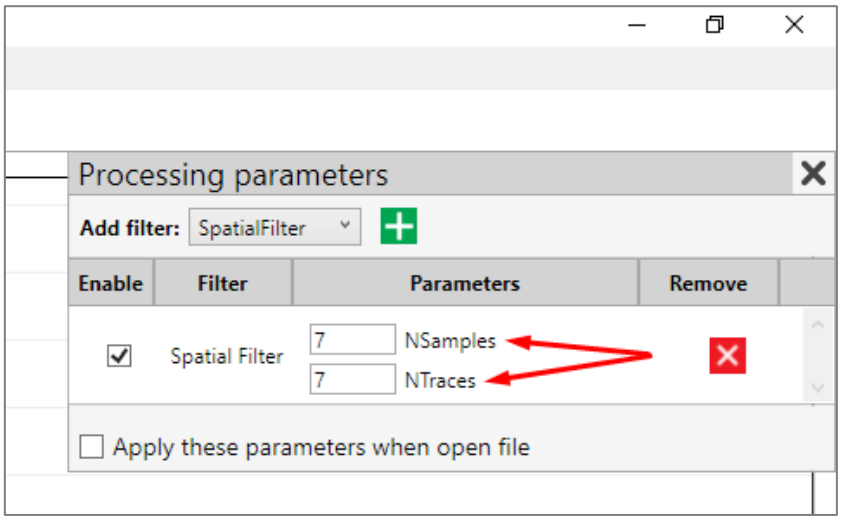

Fig. 65. Smoothing filter settings in the 2D sliding window

An example of applying the smoothing procedure in a 2D sliding window is shown in Fig. 66.

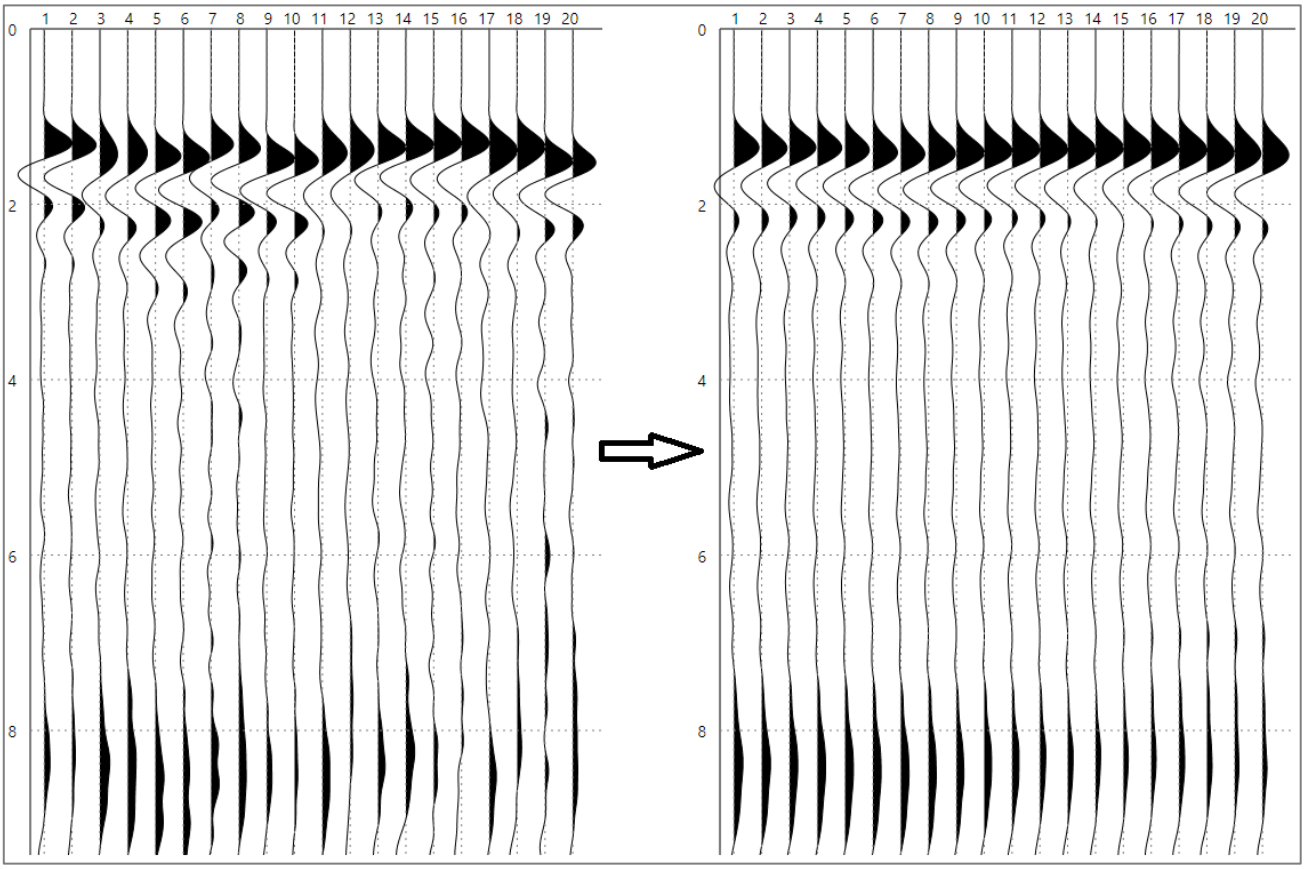

Fig. 66. Example of applying the smoothing procedure in a 2D sliding window

3.3. Signal spectrum window

To open the signal spectrum analysis window, select the seismic gather in the project tree by double-clicking the left mouse button and click the "spectrum" button in the main program window (Fig. 67).

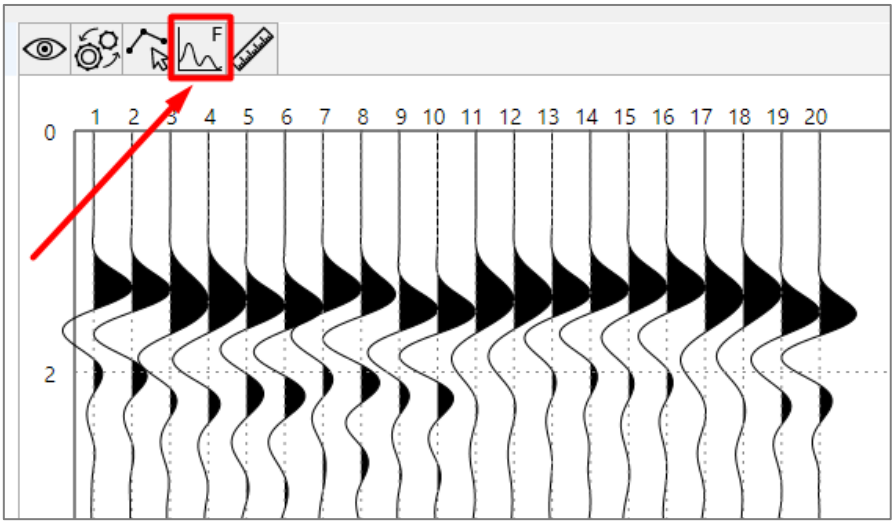

Fig. 67. Button to open the signal spectrum window

After clicking the button "Spectrum" the window with the average spectrum of the selected seismic gather will open (Fig. 68). The name of the file for which the spectrum is built is displayed in the header of the spectrum window. You can open any number of windows with spectra for one or different files.

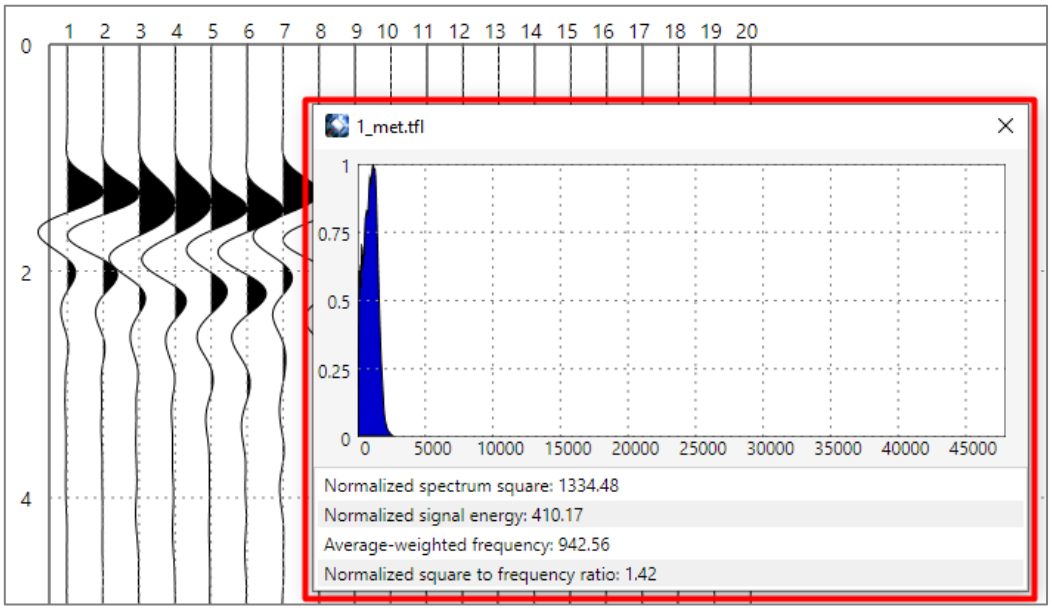

Fig. 68. Signal spectrum window

The upper part of the window shows the signal's spectrum. When you move the mouse in the spectrum display area, the frequency corresponding to the cursor position is shown (Fig. 69).

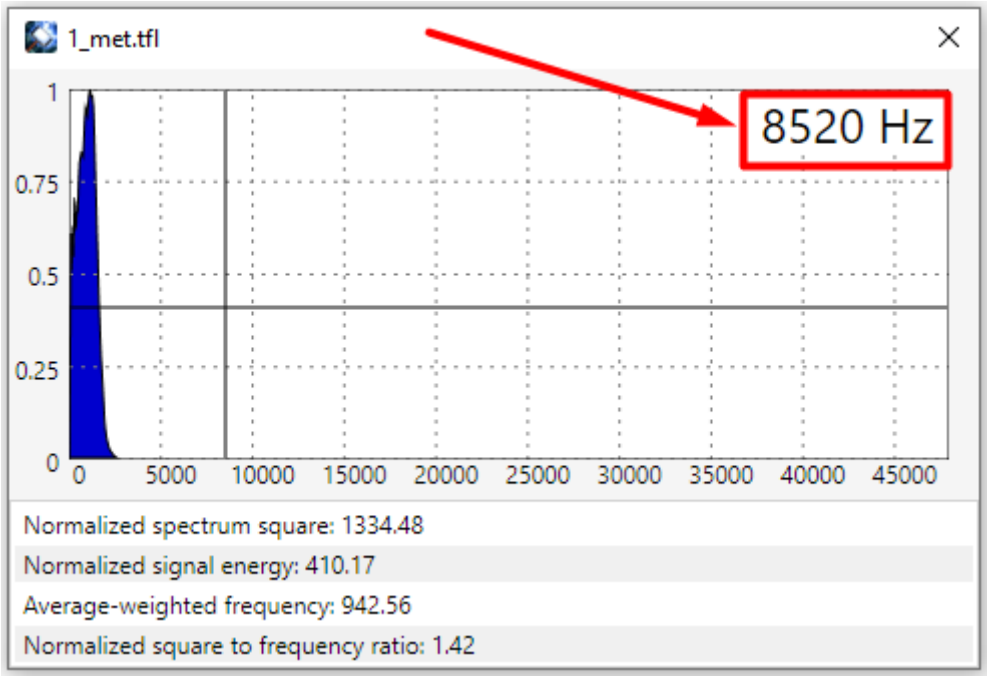

Fig. 69. Frequency value corresponding to the cursor position

When you click with the left mouse button on the spectrum display area, the upper limit of the frequency scale (horizontal axis) will change to the value corresponding to the place where you clicked with the left mouse button (Fig. 70). To reset the upper limit to the original maximum value, right-click anywhere in the signal's spectrum display area.

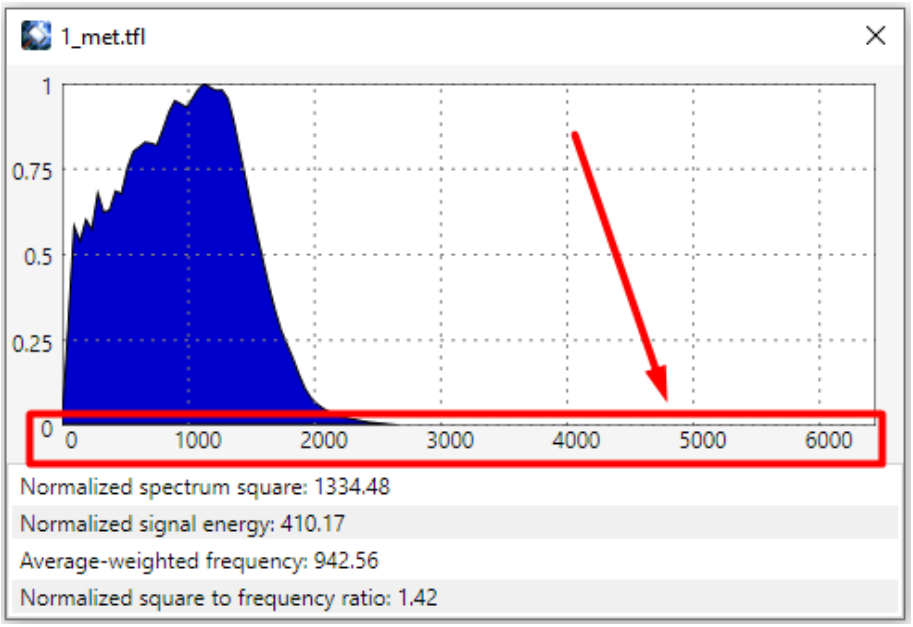

Fig. 70. Display of the signal spectrum when the upper limit of the frequency scale changes

The bottom part of the window with the signal's spectrum displays the attributes calculated from the spectrum and the signal (Fig. 71).

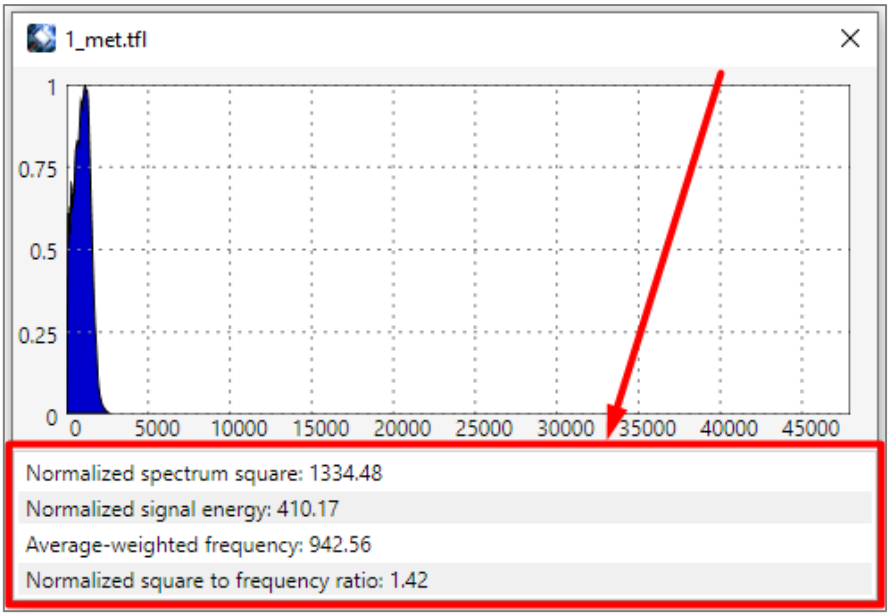

Fig. 71. Attributes calculated by spectrum and signal

The software calculates the following attributes:

1. Normalized spectrum square.
2. Normalized signal energy.
3. Average-weighted frequency.
4. Normalized square to frequency ratio.

Attributes can be used to analyze single spectrograms as well as to build attribute maps when working with the impact method (see section "Impact method (building attribute maps»).

3.4. First break picking and trace rejection

To open the picking tool, click the corresponding button of the toolbar in the main program window (Fig. 72).

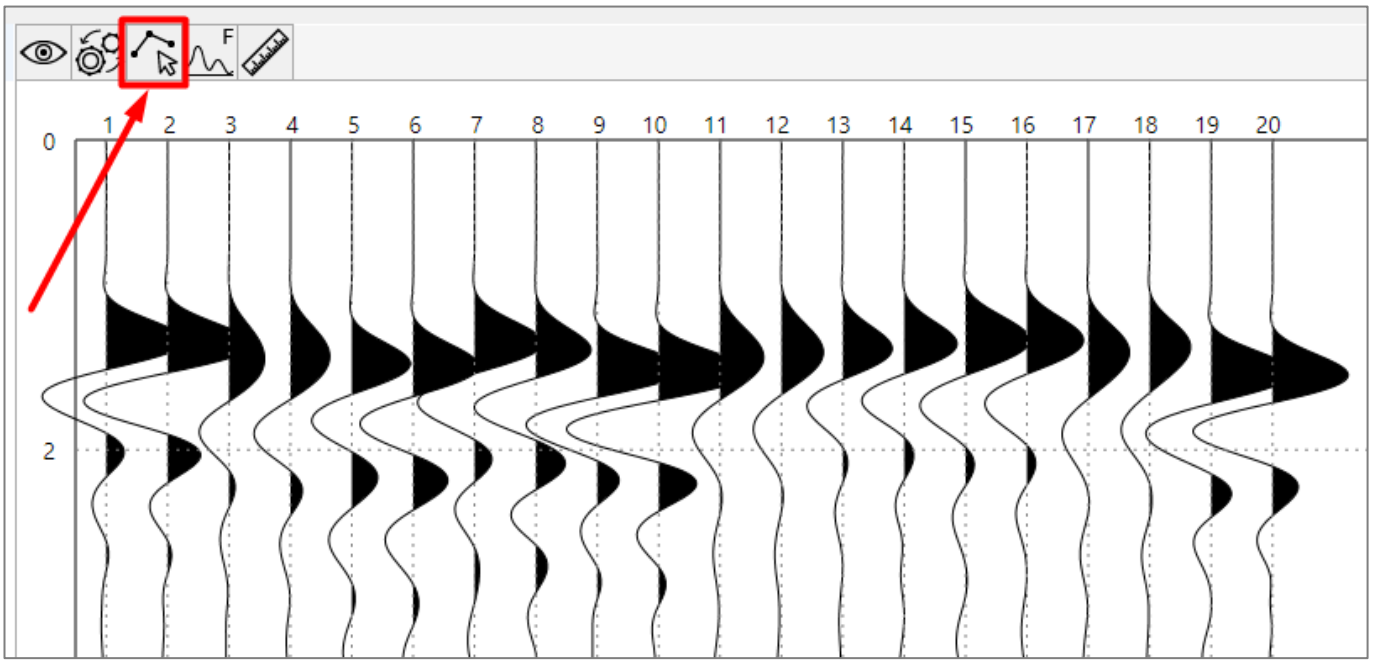

Fig. 72. Opening the picking tool

After clicking the "Picking Tool" button in the upper right corner of the main program window the corresponding interface will appear (Fig. 73).

Fig. 73. Picking Tool interface

Picking Tool allows one to perform two operations (the option is selected in the tool's list):

1. First break picking.
2. Picking of rejected traces.

3.4.1. First break picking

Select the HandStaticMode option in the list to switch to the first break picking (Fig. 74).

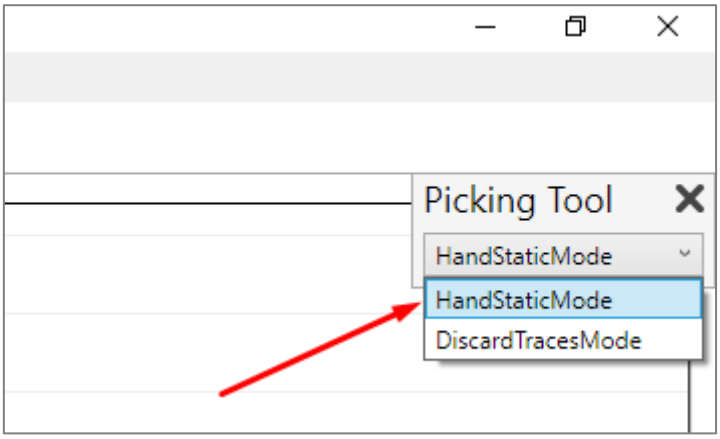

Fig. 74. Selecting the first break picking mode

Mark the first break for each seismic gather trace by dragging the pick marker with the left mouse button pressed (Fig. 75).

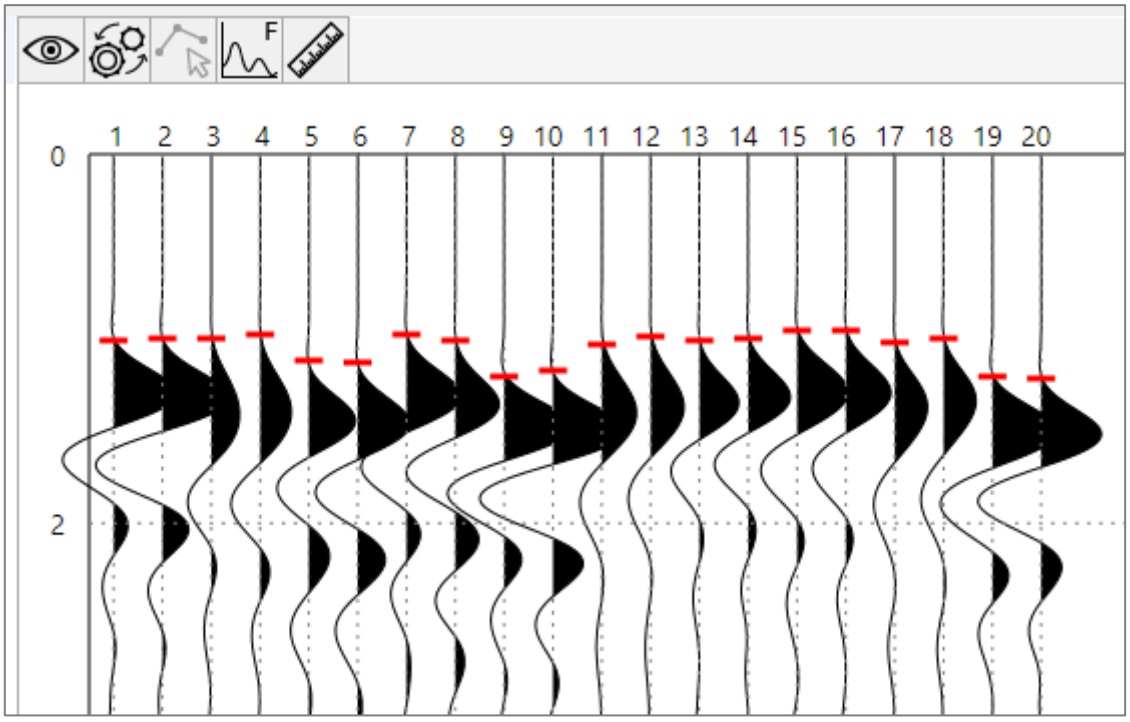

Fig. 75. First break picking

3.4.2. Trace rejection

To switch to DiscardTracesMode, select the corresponding option in the (Fig. 76).

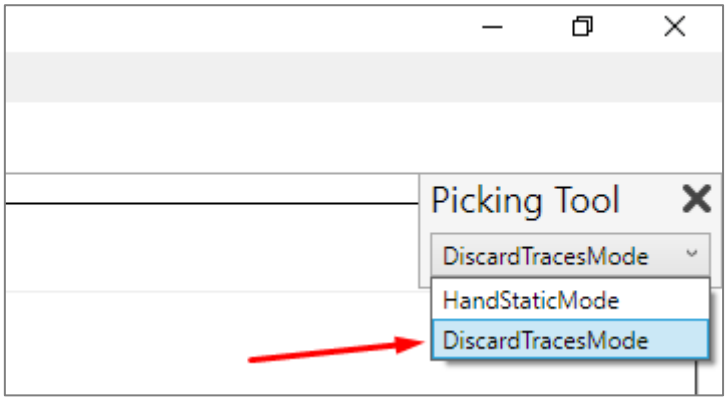

Fig. 76. Selecting the trace rejection mode

Click the left mouse button to mark the rejected traces. The selected traces are marked with red rectangles (Fig. 77).

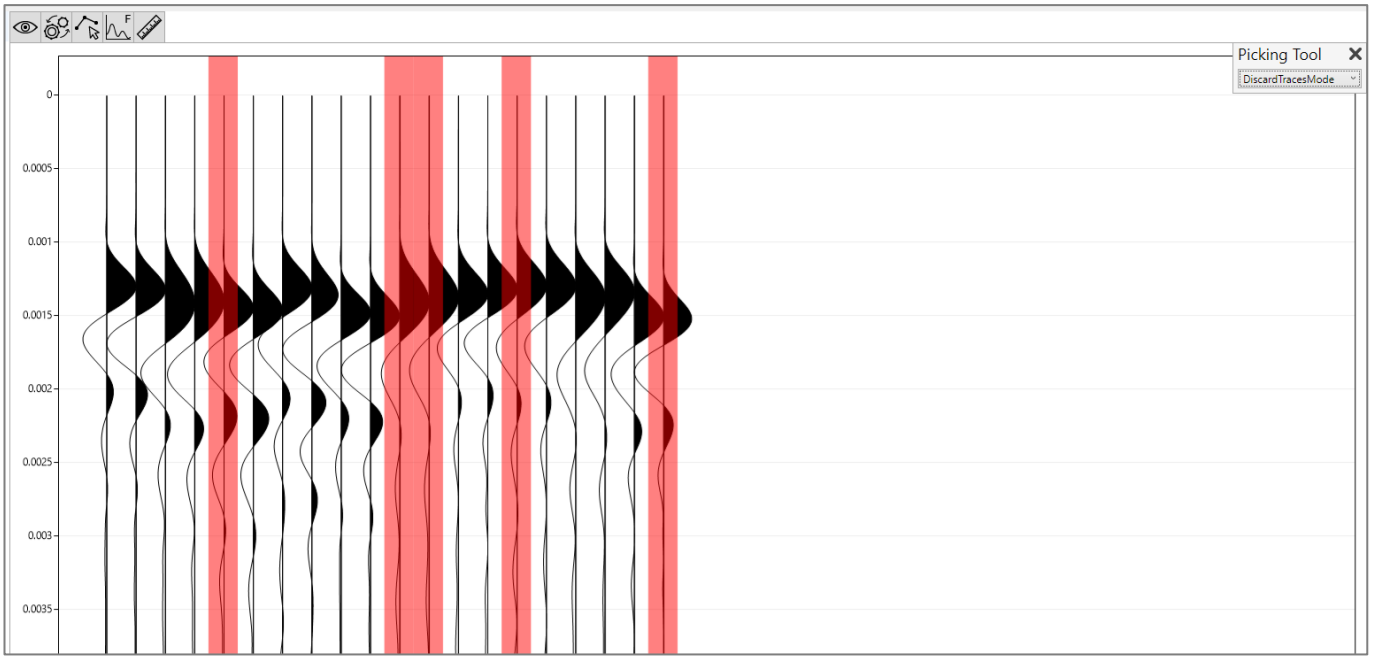

Fig. 77. Examples of marked rejected traces.

4 INTERPRETATION (PILE CONTROL)

4.1. Панель интерпретации сейсмограммы

The seismic gather interpretation panel is opened by the button "ruler" on the toolbar in the main program window (Fig. 78).

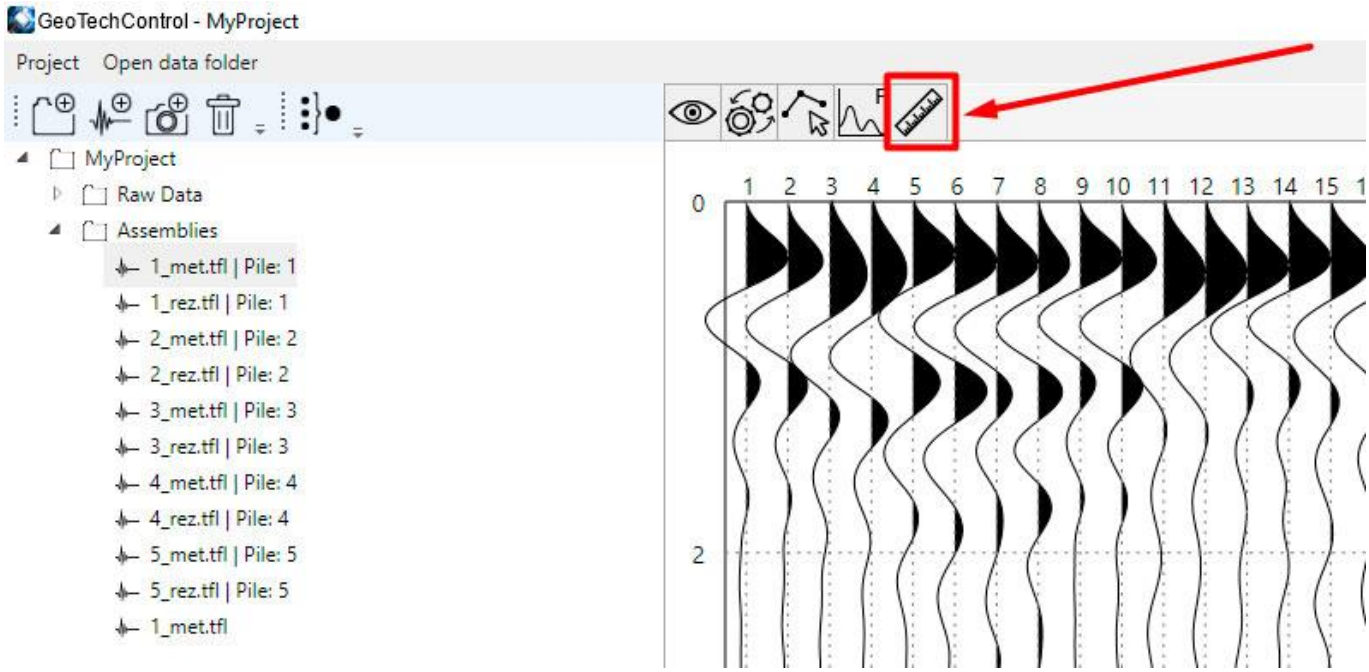

Fig. 78. Opening the seismic gather interpretation panel

After clicking on the button of the interpretation panel it will open (Fig. 79).

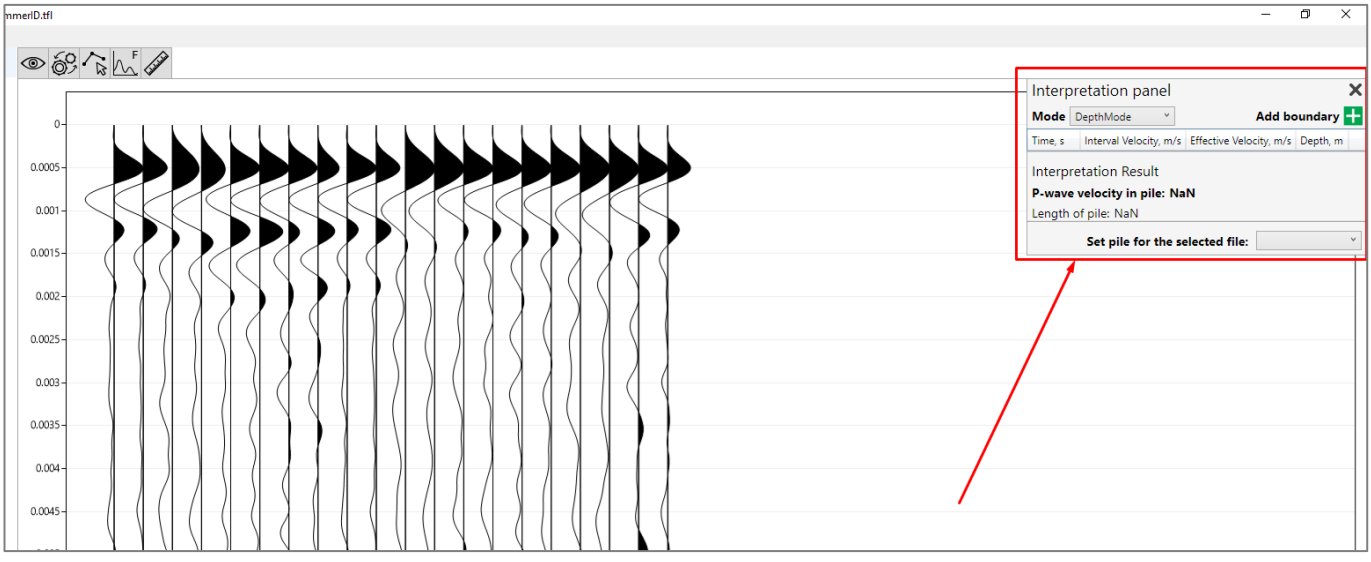

Fig. 79. Seismic gather interpretation panel

Interpretation is available in two modes (Fig. 80):

1. Velocity mode. In this mode user enters an interval velocity for the boundaries, based on which the depths (lengths) are calculated.
2. Depth mode. In this mode user enters the boundaries' depths based on which the velocities are calculated.

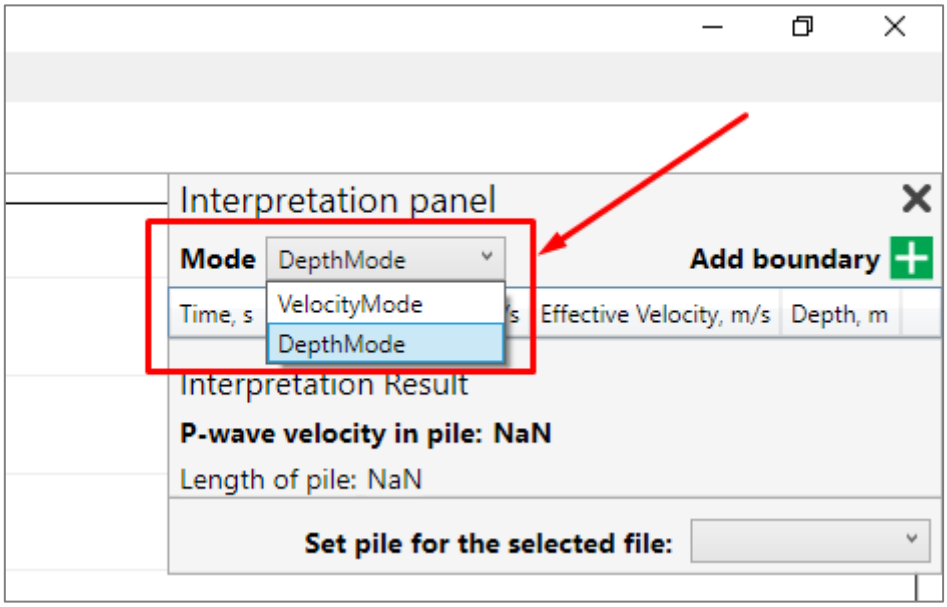

Fig. 80. Selecting the interpretation mode

To add a boundary, click on the corresponding button (Fig. 81). The first boundary is added without velocity and with a depth equal to 0 m - it is the top of the pile.

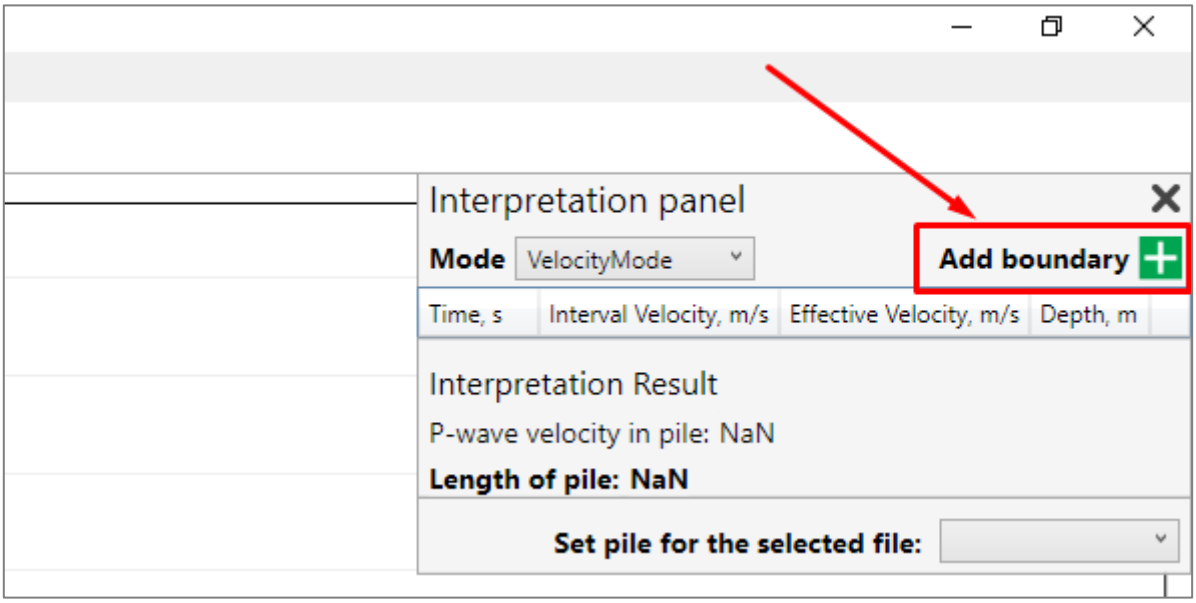

Fig. 81. Adding the boundary

Each added boundary appears (Fig. 82):

1. In the table on the interpretation panel: each row of the table represents a boundary.
2. As a horizontal blue line on the seismic gather.

You can change the position of the boundary by moving the corresponding horizontal blue line with the mouse holding the left button. After moving the boundaries you have to update the table in the interpretation panel by clicking the "update" button (Fig. 82).
The bottom part of the interpretation panel will show the interpretation result (Interpretation Result – Fig. 82). The result of either pile length or effective velocity in the pile is marked in bold, depending on the interpretation mode selected.

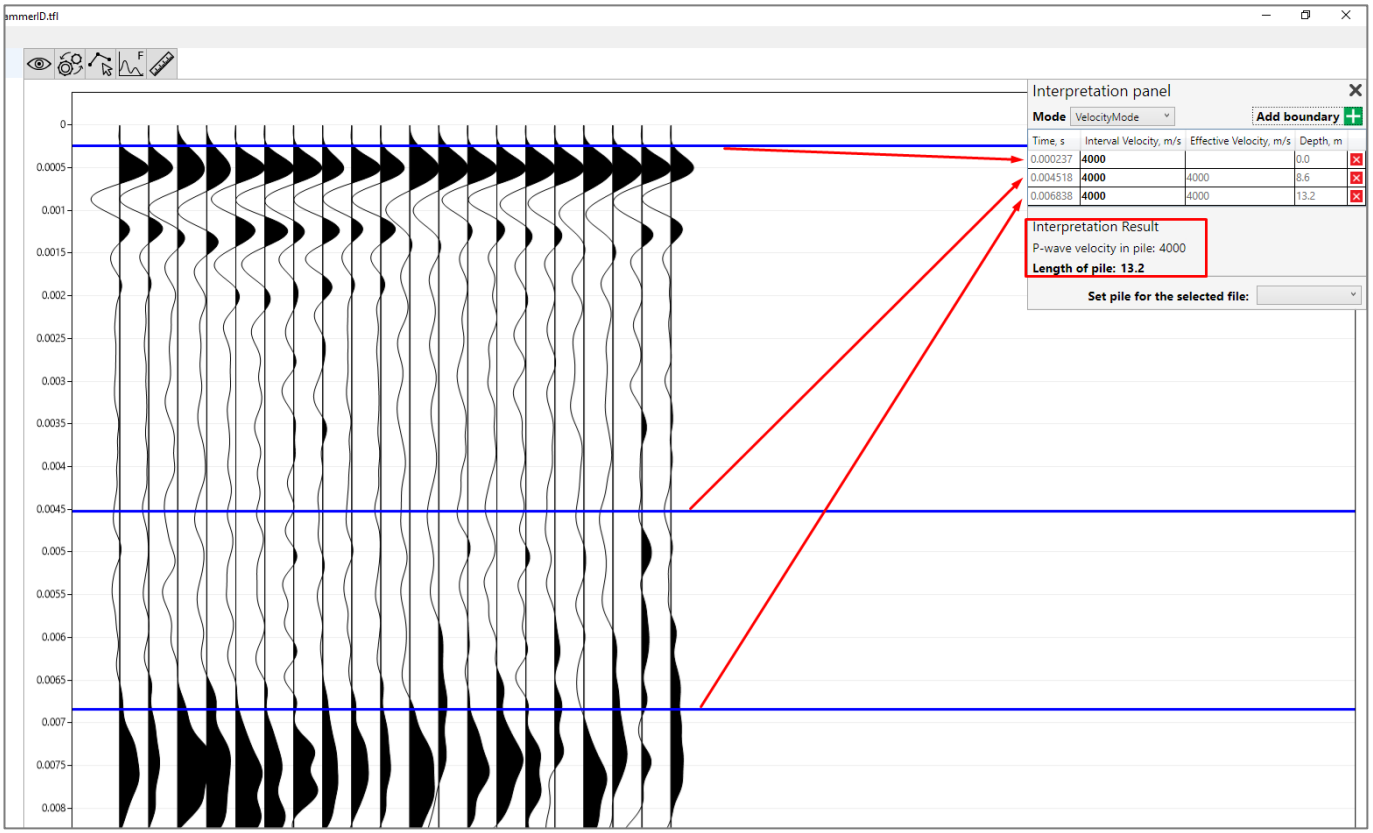

Fig. 82. Added boundaries and the result of pile length calculation

In the interpretation table in "Velocity mode" you can edit the interval velocity (Fig. 83). In the depth mode you can edit the depth for each boundary.

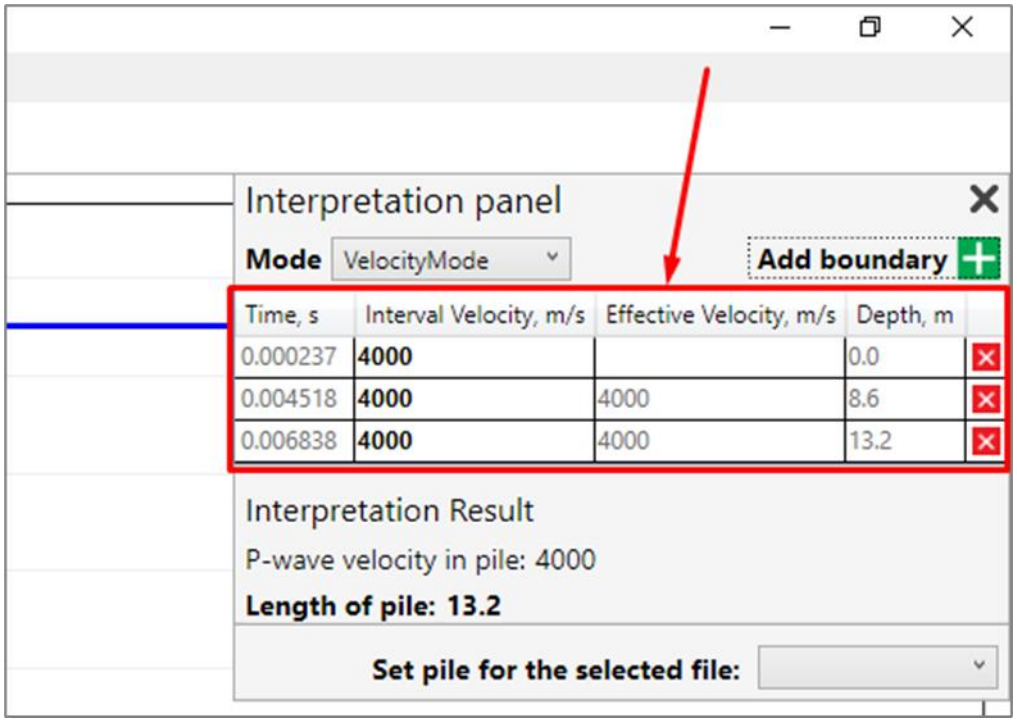

Fig. 83. Interpretation table for seismic gather

To remove the boundary, click the "X" button. (Fig. 84).

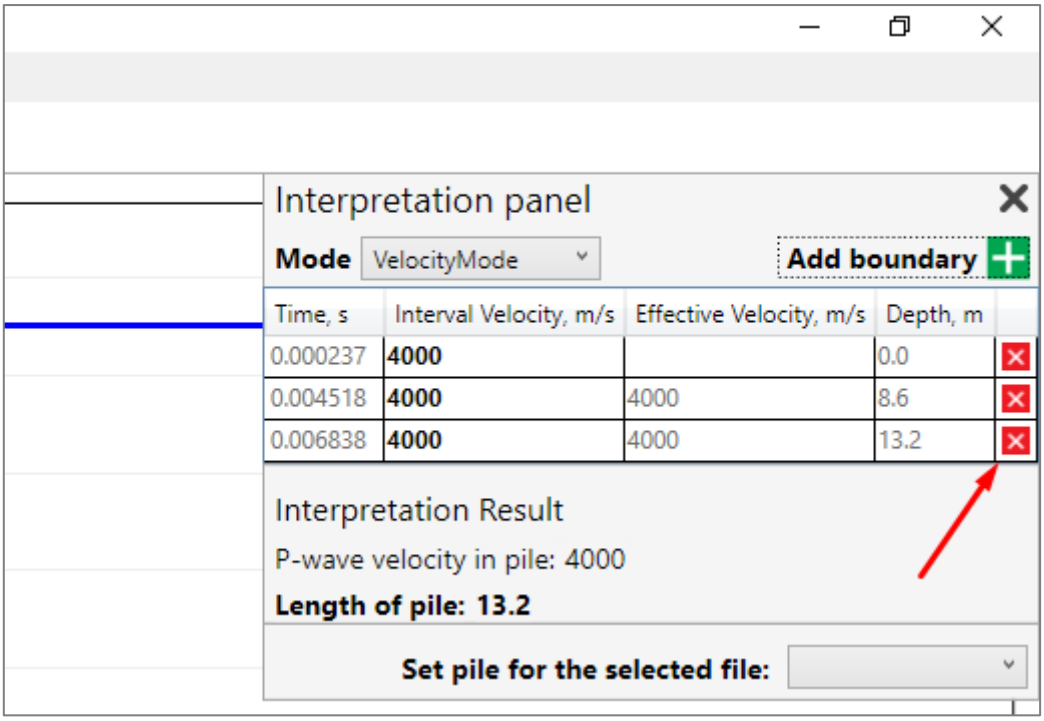

Fig. 84. Button for deleting boundary

To link the seismic gather to a pile, you need to select the required pile in the list of piles at the bottom of the interpretation panel (Fig. 85). If there are no piles in the list, you need to add them in the project settings.

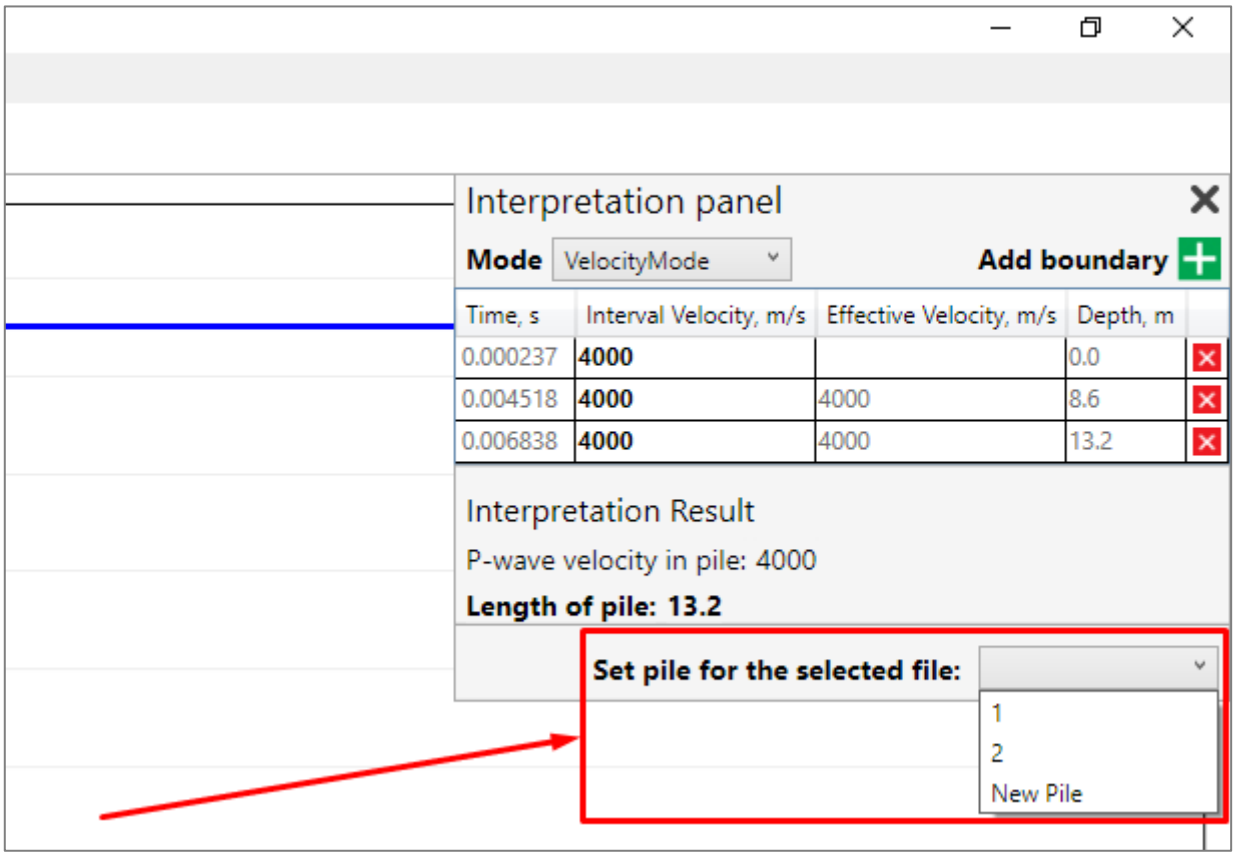

Fig. 85. Selecting a pile in the list of piles in the interpretation panel

4.2. Graphical report building based on pile

In order to build a graphical report, go to menu "Project->Export->All pile reports" menu (Fig. 86).

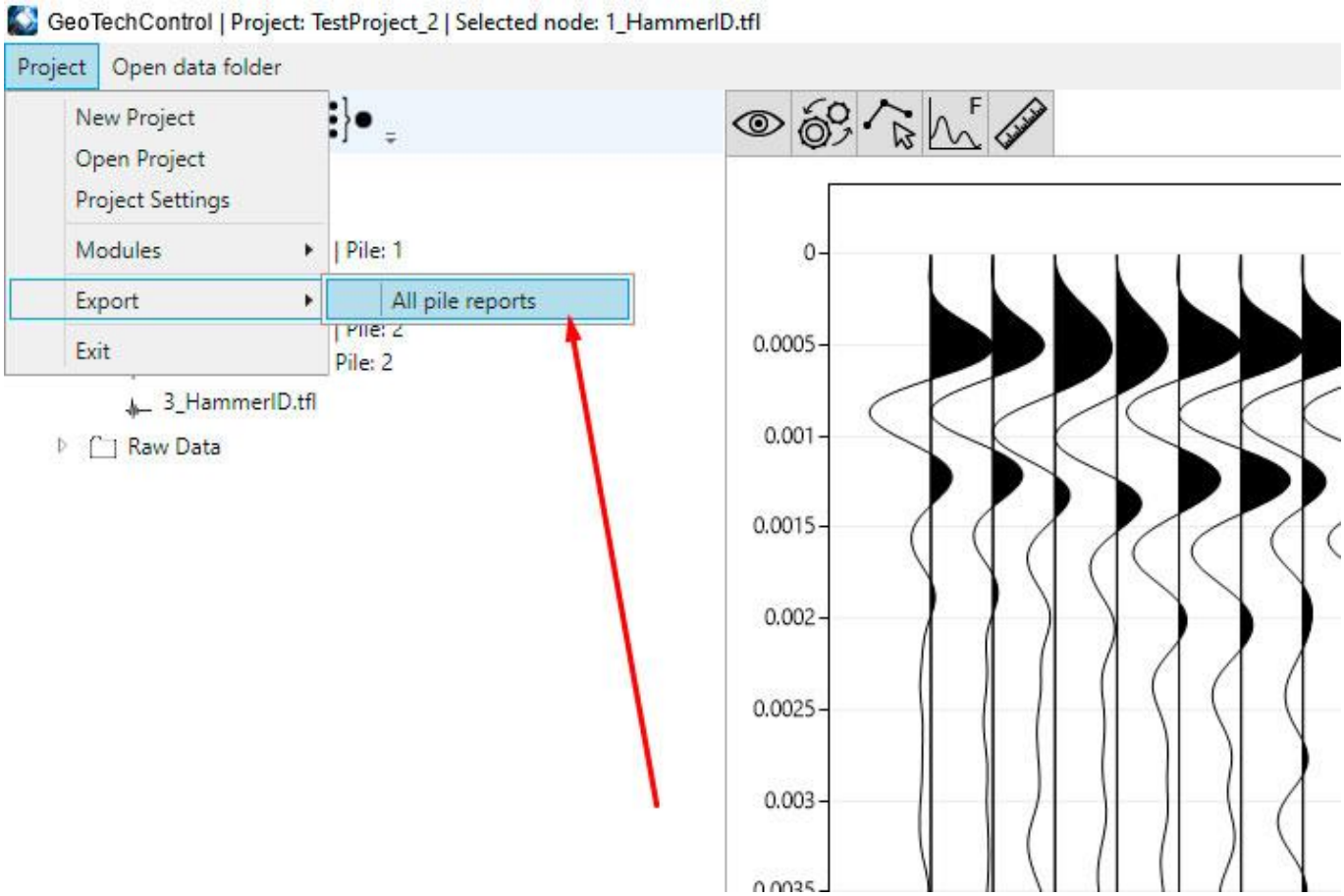

Fig. 86. Menu for graphical reports export

Reports for all files that have an interpretation and are linked to piles will be generated automatically and saved as ".png" images in the folder specified in the folder selection dialog box. An example of graphical report is shown in (Fig. 87).

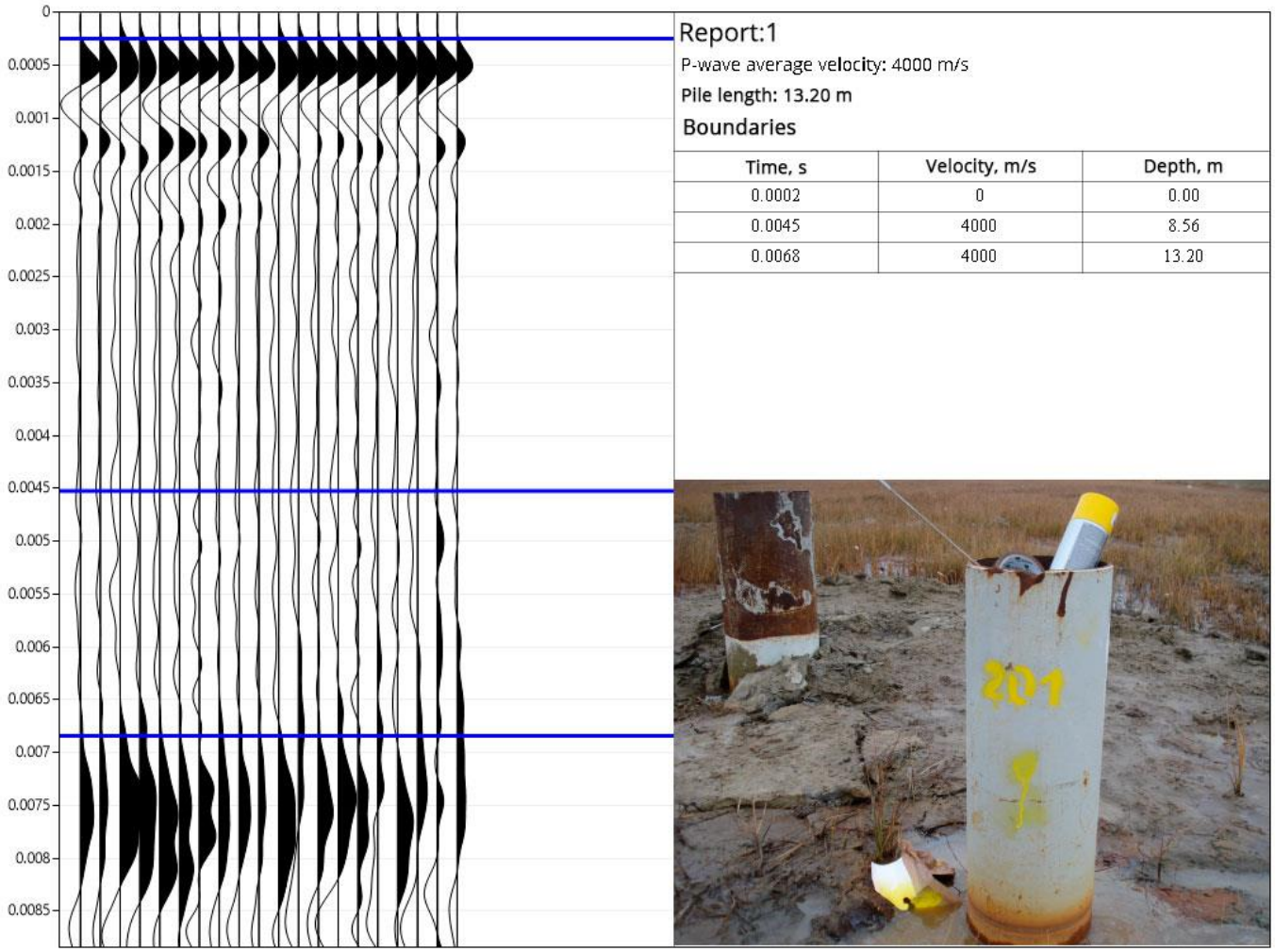

Fig. 87. Example of a pile report

5. IMPACT METHOD (BUILDING ATTRIBUTE MAPS)

Pile-MASTER software implements a tool that allows one to work via impact method. The tool enables to process areal data and build attribute maps calculated from the spectrum of recorded signals.
To open the Impact Method tool, use the "Project-Impact Method" menu" (Fig. 88).

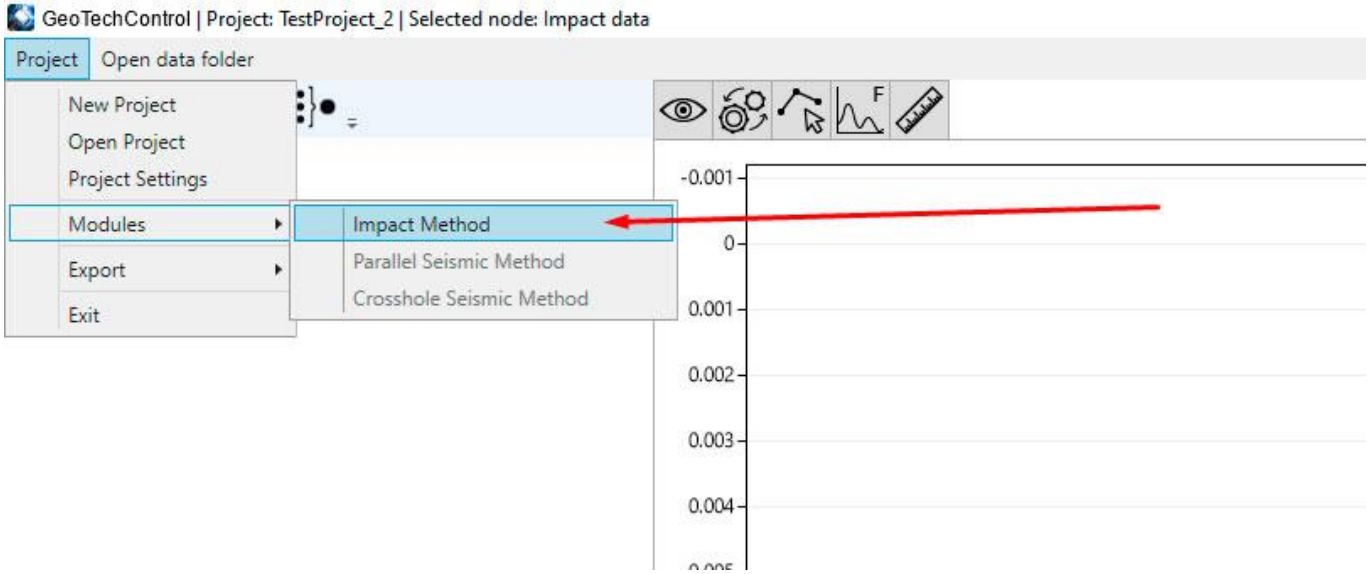

Fig. 88. Opening the Impact Method tool

5.1. Preparing data for building an attribute map

To build an attribute map, the areal survey raw data must be loaded into the project (Fig. 89).

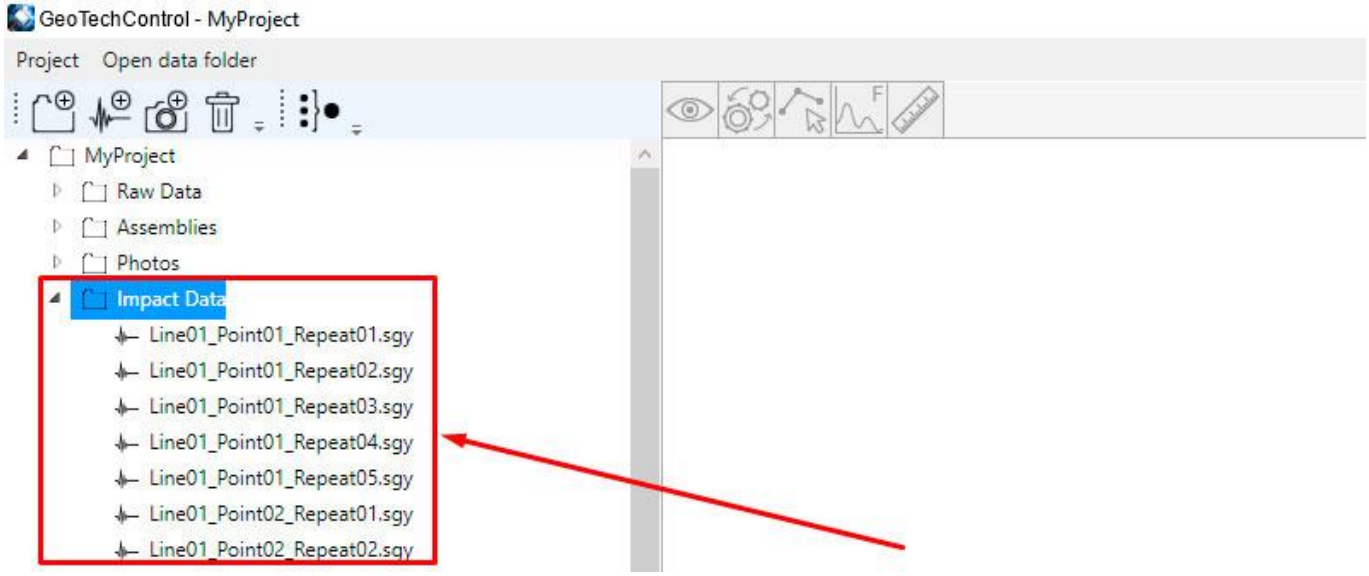

Fig. 89. Areal survey data loaded into the project

Open the "Project-Modules-Impact Method" menu for call-in of the impact method window. Fig. 90 shows the general view of the window. In the left part of the window the control tools are located, in the right part of the window there is an area of map building.

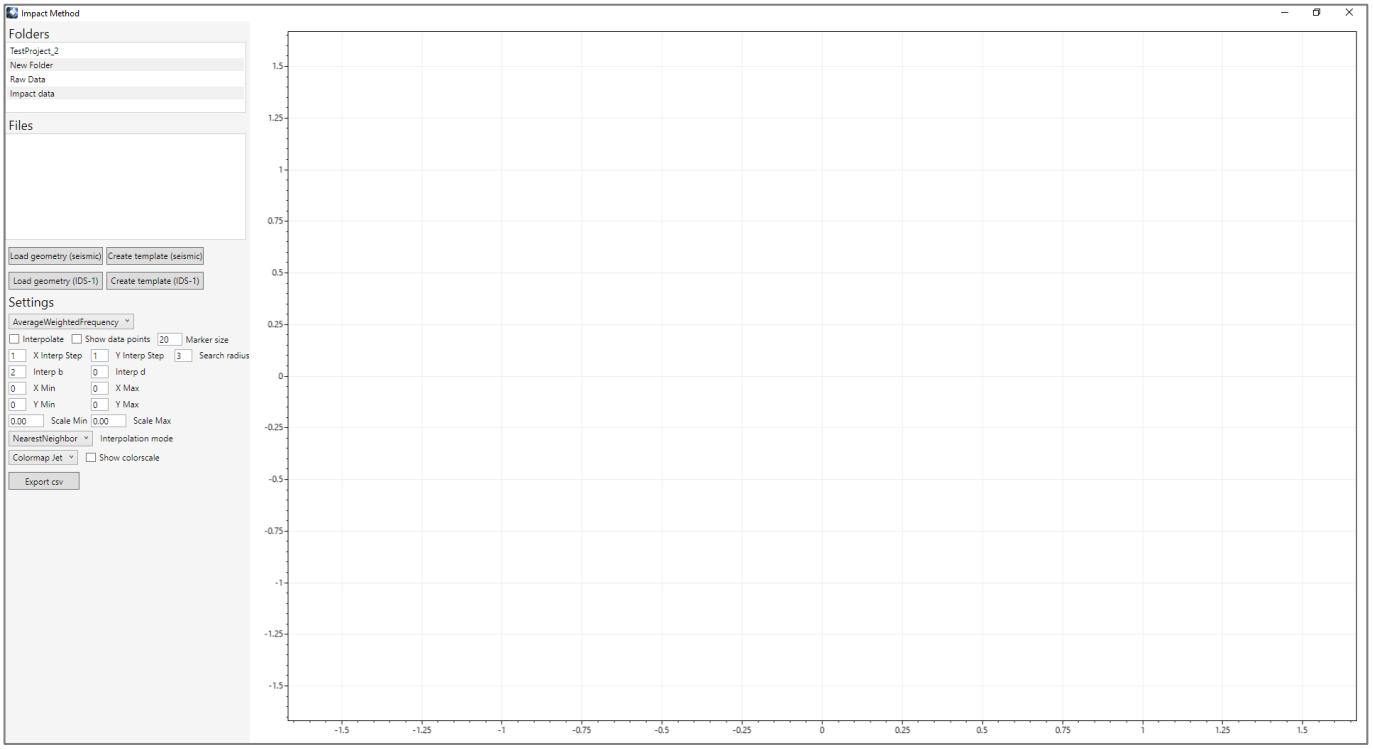

Fig. 90. Impact method window

The "Folders" list shows the folders available in the project (Fig. 91).

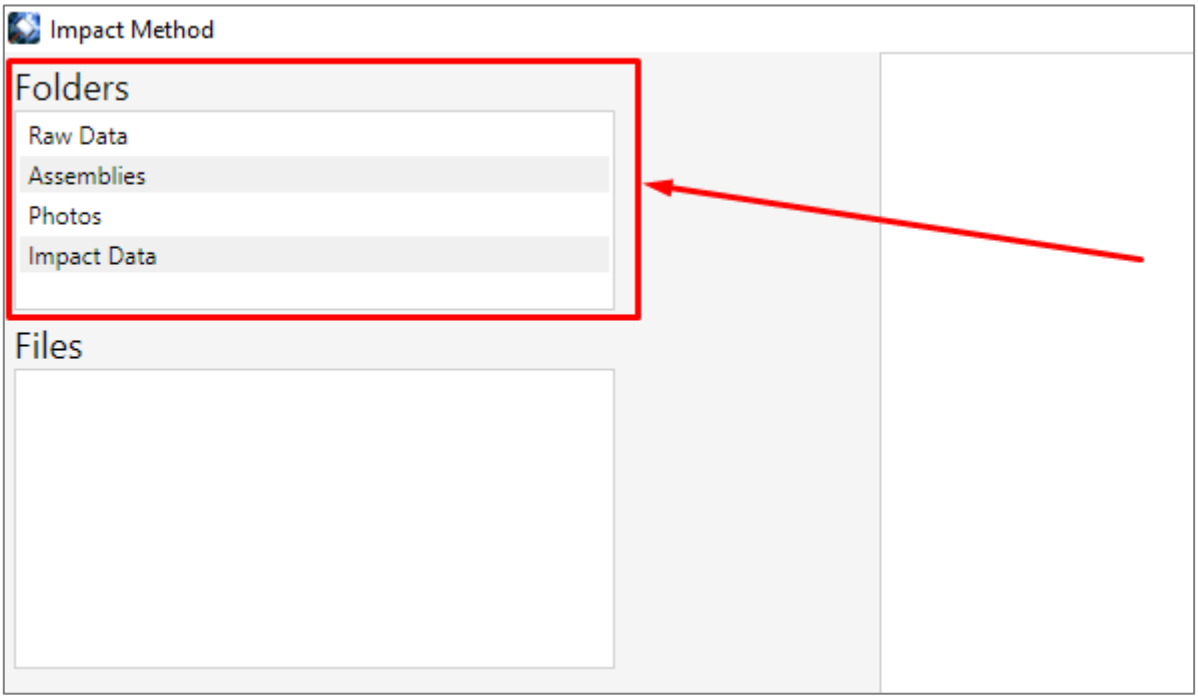

Fig. 91. List of project folders in the Impact Method window

Select the required folder in the list of folders – with the loaded areal survey data. The list of files of the selected folder will be displayed in corresponding list (Fig. 92).

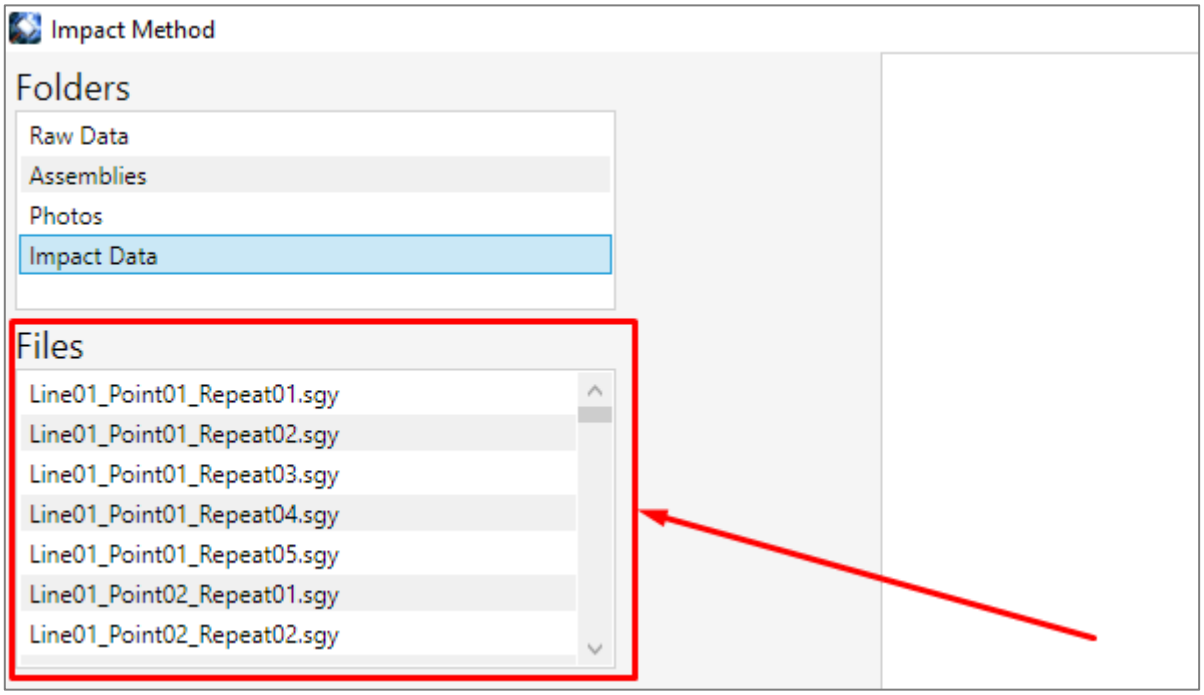

Fig. 92. File list of the folder selected in the "Folders" list

File names must be organized in such a way that they contain information about the observation point and repetition number. An example of the correct organization of file names is shown in Fig. 92. In the name (from left to right) there should be written: profile number – point number – repetition number.
Then one needs to prepare and upload a geometry file, after which the attribute map building will be available.

5.1.1. Preparing a geometry file in manual mode

Option 1

If the names of the raw data files are organized so that they include a profile, a point, and realization number, then use this method.
Fig. 93 shows an example of how the geometry file should be organized to be loaded into the GeoTechControl software. The file format is .csv. The sequence of columns is strictly as specified in the example.

```
1  LineId;PointId;XPosition;YPosition;Zposition;FileNameMask;Channel
2  1;1;0;0;0;Line01_Point01;all
3  1;2;1;0;0;Line01_Point02;all
4  1;3;2;0;0;Line01_Point03;all
5  1;4;3;0;0;Line01_Point04;all
6  1;5;5;0;0;Line01_Point05;all
7  1;6;7;0;0;Line01_Point06;all
8  1;7;10;0;0;Line01_Point07;all
9  2;1;0;1;0;Line02_Point01;1
10 2;2;1;1;0;Line02_Point02;1
11 2;3;2;1;0;Line02_Point03;1
12 2;4;3;1;0;Line02_Point04;1
```

Fig. 93. Geometry file example

The first line in the file represents columns headers. Each subsequent line represents one observation point.

For each observation point in the file it should be written the following:

1. Number of profile (LineId).
2. Point number on the profile (PointId).
3. X coordinate of the observation point (XPosition).
4. Y coordinate of the observation point (YPosition).
5. Z coordinate of the observation point (ZPosition).
6. Filename mask for files, written on the specific observation point (FileNameMask).
7. Channel to be taken from files to calculate attributes (Channel).

Option 2

If the names of files with raw data represent a position number (as, for example, it is implemented in the seismograph IDS-1), then use this method.

Fig. 94 shows an example of how the geometry file should be organized to be loaded into the GeoTechControl software. The file format is .csv. The sequence of columns is strictly as specified in the example.

```
1  June 30th;;;;;;
2  №;FileFrom;FileTo;X;Y;Source;Channel
3  1;1;5;6610;3;large metal;1
4  2;6;10;6610;4;large metal;1
5  3;11;15;6615;3;large metal;1
6  4;16;20;6615;4;large metal;1
7  5;21;25;6620;3;large metal;1
8  6;26;30;6620;4;large metal;1
9  7;31;35;6625;3;large metal;1
```

Fig. 94. Geometry file example

The first line in the file is a comment.

The second line in the file represents columns headers. Each subsequent line is one observation point.

For each observation point in the file it should be written the following:

1. Number of point (№).
2. Number of the first file on the point (FileFrom).
3. Number of the last file on the point (FileTo).
4. X coordinate of the observation point (XPosition).
5. Y coordinate of the observation point (YPosition).
6. Type of source (Source).
7. Channel to be taken from files to calculate attributes (Channel).

When initializing geometry through both the first and the second method, it is necessary to keep in mind:

- The point position must be specified in a rectangular coordinate system.

- Channel is specified by a value corresponding to the number of the channel to be taken for the calculation from files. If you want to perform calculations for all channels from the file, you should write "all" in the "Channel" column.

- The filename mask for files must be the common initial part of the file names related to this observation point. For example, if the names of the files recorded at a point are given as "Line01_Point01_Repeat01" and "Line01_Point01_Repeat02", then the filename mask can be "Line01_Point01_Repeat" or "Line01_Point01". That is, the repetition number is not included in the mask, but the point and profile number are included in the mask, so that the set for a given point contains all the required repetitions.

5.1.2. Preparing a geometry file via template generation

After selecting the folder with signals in the impact method window, you can generate a template of the geometry file (Fig. 95).
The program will automatically generate a geometry file template for the selected set of names of files with signals. The masks will be defined with the condition that the repetition number is the last characters in the file name.

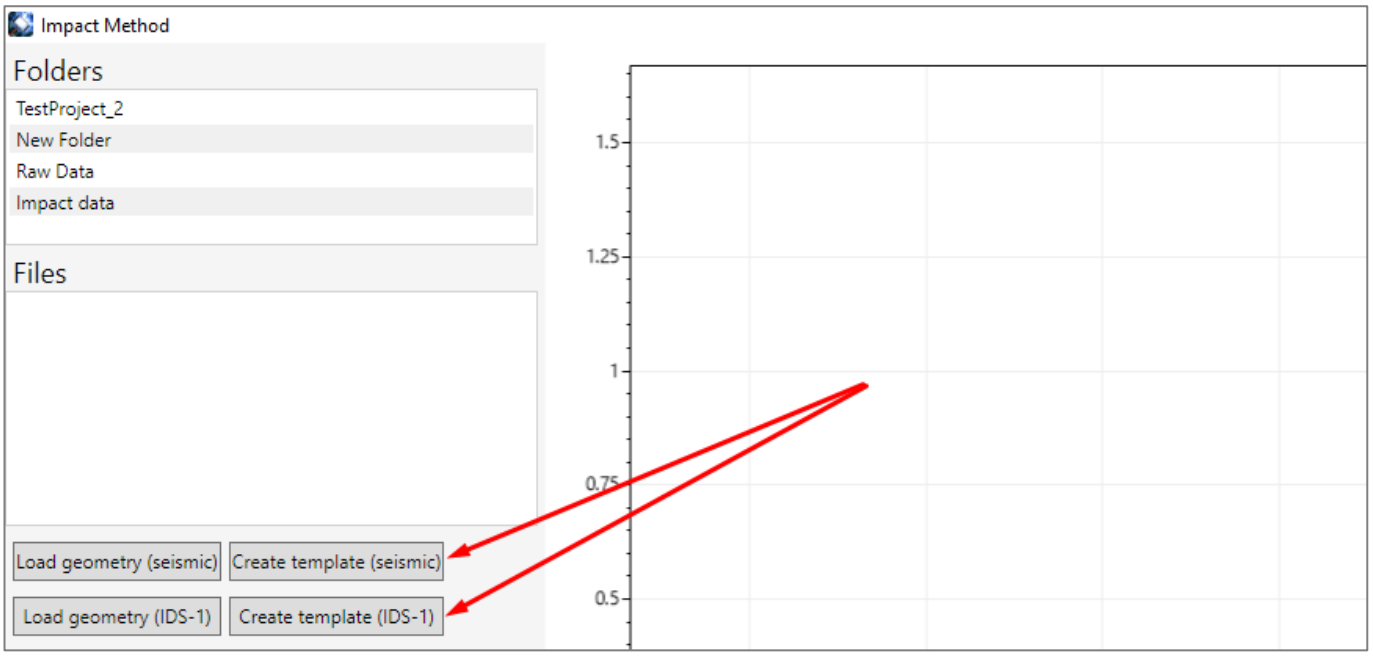

Fig. 95. Button for generating a geometry file template

In the appearing save file dialog box enter the name of the template file and click "save".

As a result, a geometry file will be generated based on the set of files displayed in the "Files" list. In the name of such file masks will be written and it will be necessary to add numbers of profiles, observation points, point coordinates and channel (Fig. 96).

```
Line Id;Point Id;Xposition;Yposition;ZPosition;Filename mask;Channel;
;;;;;Line01_Point01_Repeat;;
;;;;;Line01_Point02_Repeat;;
;;;;;Line01_Point03_Repeat;;
;;;;;Line01_Point04_Repeat;;
;;;;;Line01_Point05_Repeat;;
;;;;;Line01_Point06_Repeat;;
;;;;;Line01_Point07_Repeat;;
;;;;;Line02_Point01_Repeat;;
;;;;;Line02_Point02_Repeat;;
```

Fig. 96. Example of a generated geometry file template (seismic mode)

Fig. 97 shows an example of the generated geometry file template in "IDS-1" mode – in the case when the file names are numbers.

testTemplateIDS.csv

```
Geometry file for IDS-1 (EXAMPLE)
LineId;File From;File To;XPosition;YPosition;Zposition;Channel
1;1;10;1;0;0;1
1;11;20;2;1;0;1
```

Fig. 97 Example of a generated geometry file template (IDS-1 mode)

5.2. Building attributes map

Before attributes map building select the folder with impact method data in the "Folders" list and load to the program the geometry file, prepared according to the above described requirements (see section 5.1.1 and section 5.1.2) and corresponding to the selected set of files. To load a geometry file by the first method (see section 5.1.1 – Option 1) press the “Load geometry” button. To load a geometry file by the second method (see. section 5.1.1 – Option 2) press the “Load geometry IDS-1” button.

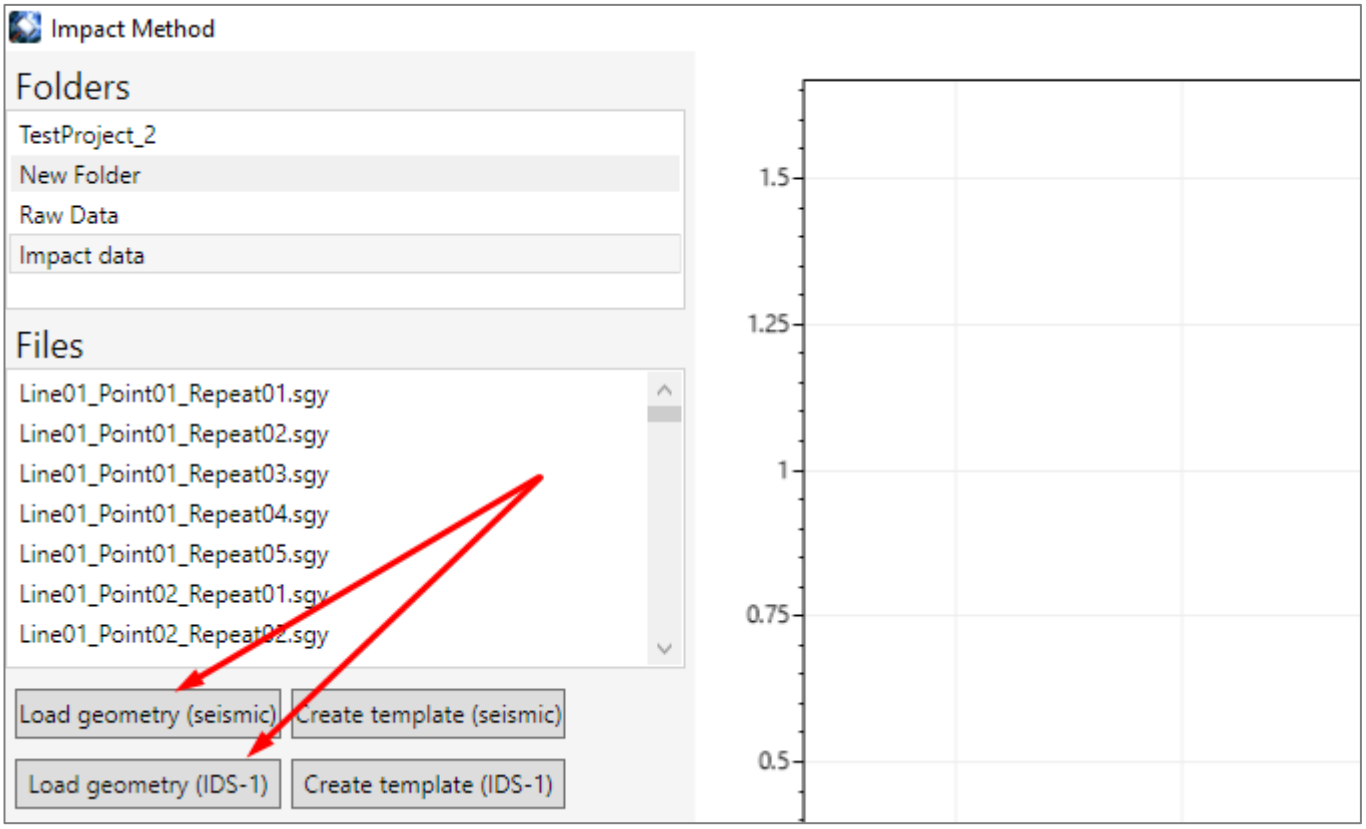

Fig. 98. Импорт файла геометрии в программу

After the geometry file has been successfully loaded, the limits of the attributes values (color scale) will be updated in the lower part of the settings panel in the Impact Method window (Fig. 99):

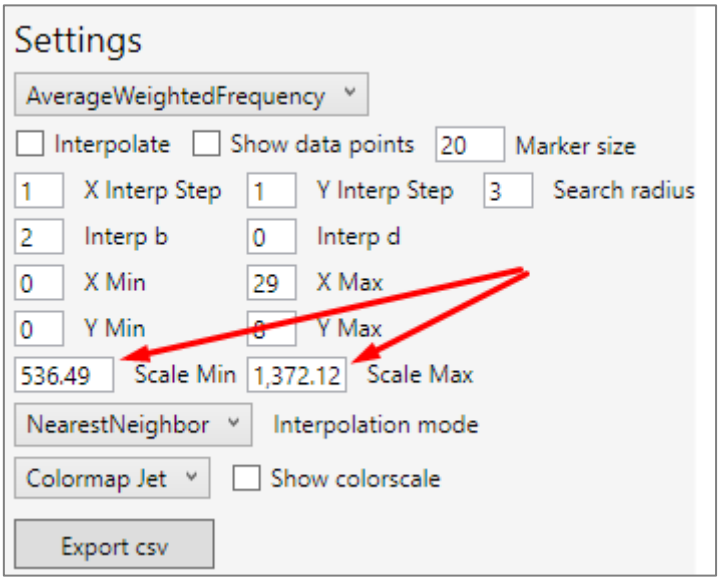

Fig. 99. Обновленные значения пределов карты и атрибута после успешной загрузки геометрии

In the "Settings" panel, by default, a non-interpolated attribute map will be built, showing the data points with shaded squares, and the attribute value with color (Fig. 100).

Fig. 100. An example of a built non-interpolated attribute map

5.3. Parameters for building and displaying the attributes map

5.3.1. Setting of building and displaying a map

Parameters of map building and displaying are adjusted in the "Settings" block of the control panel of the Impact Method window (Fig. 101).

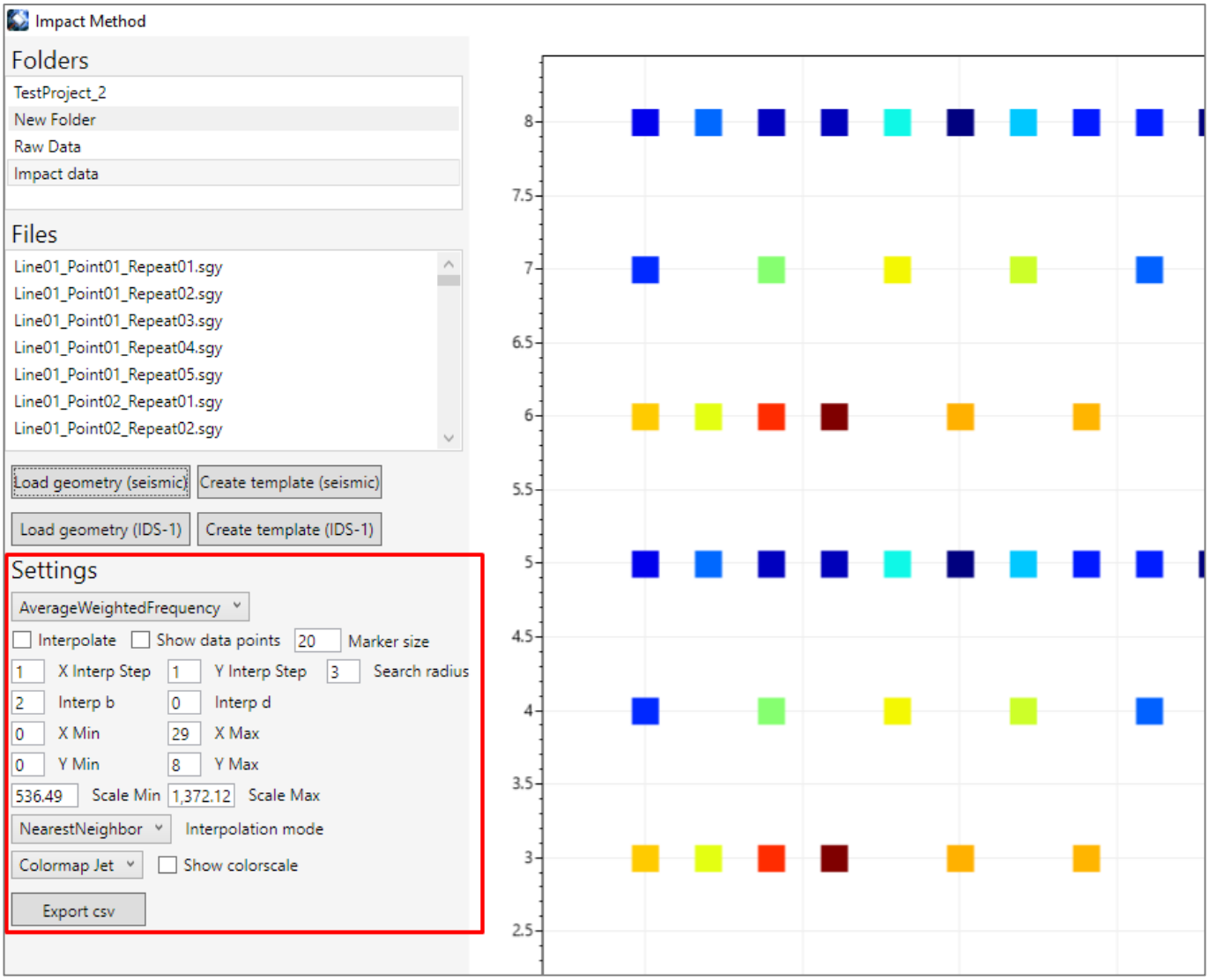

Fig. 101. Control tools for building and displaying the attribute map

For map building the following 5 attributes are available (Fig. 102):

1. Average-weighted frequency.
2. Normalized signal energy.
3. Normalized spectrum square.
4. Normalized spectrum energy.
5. Normalized square to frequency ratio.

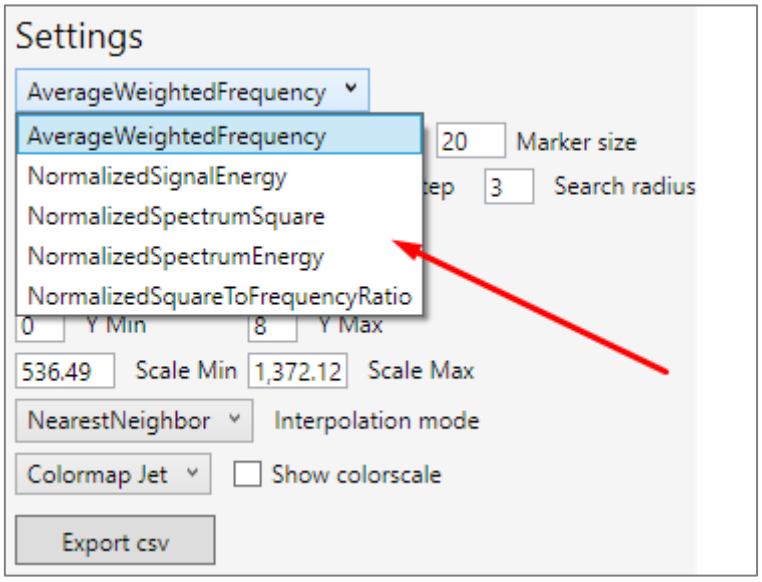

Fig. 102. Selecting an attribute to build a map

To change the color palette, select one from the list (Fig. 103).

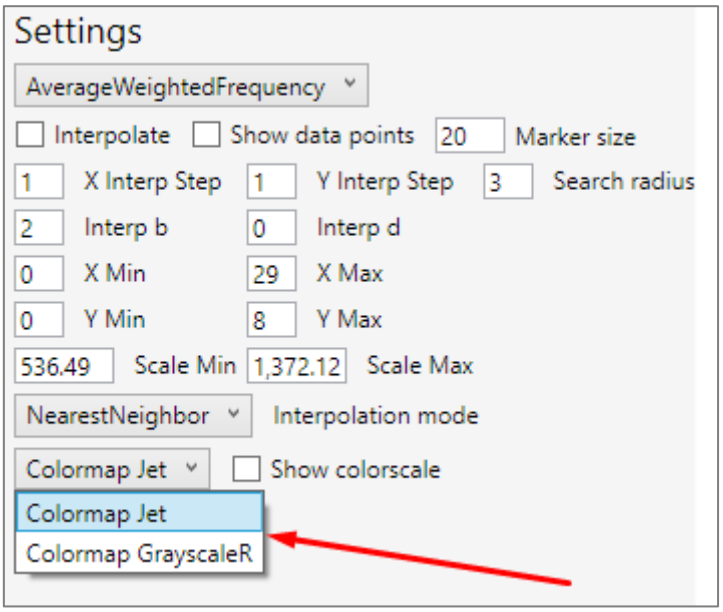

Fig. 103. Button to open the color palette setup window

The limits of the attribute’s displayed values are set by the parameters "Scale Min" and "Scale Max" (Fig. 104).

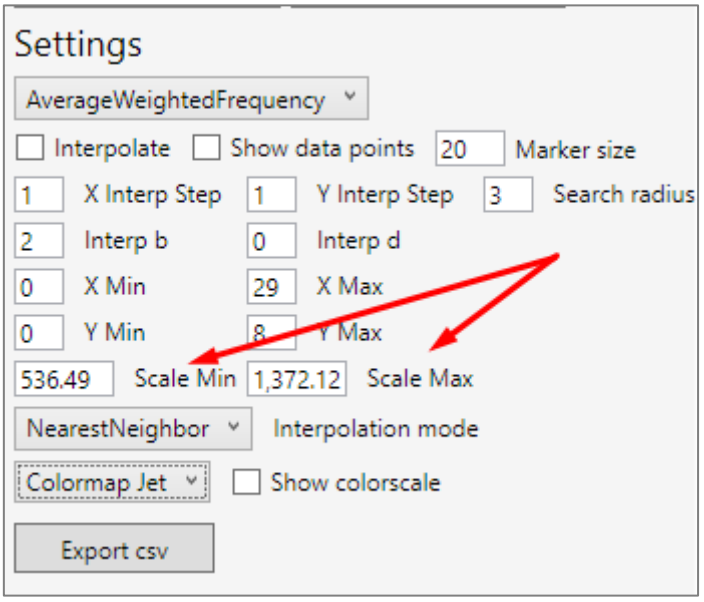

Fig. 104. Controlling the minimum and maximum values of the attribute scale

In addition to the above, the software allows one to control a number of other parameters (Fig. 105).

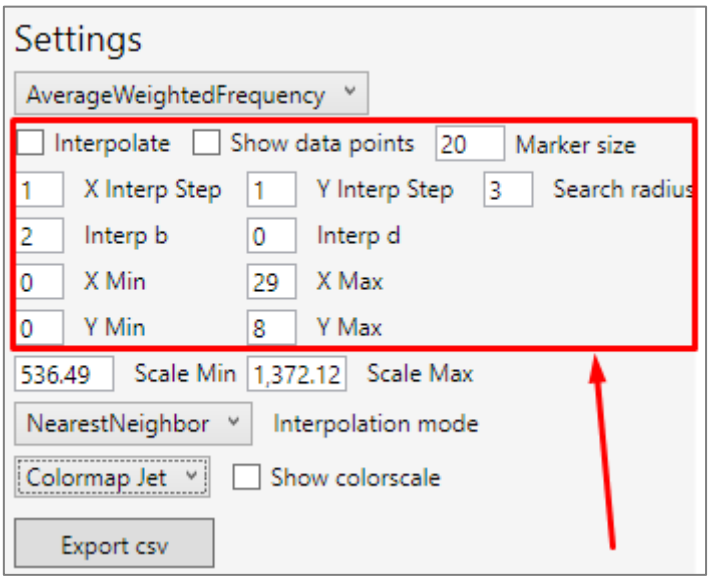

Fig. 105. Control panel of map building parameters

For settings, the following parameters are available:

1. Marker Size – the size of the squares (in pixels) showing the measurement points when drawing the non-interpolated map.
2. Show Data Points – flag that sets whether or not to show the data points from which the map is drawn. The data points are represented by round markers on the map.
3. Interpolate –flag that sets whether or not to interpolate the data for building a map.
4. X Interp Step –interpolation step along the X-axis in meters (applies if the Interpolate flag is enabled).
5. Y Interp Step –interpolation step along the Y-axis in meters (applies if the Interpolate flag is enabled).
6. Search Radius – radius of the circle in which the data points are searched in order to calculate the interpolated value. All data points that fall within a circle of a given radius will be

considered when calculating the interpolated value with weights corresponding to the distance from the interpolation point.

7. Inerp b – attenuation of the data point weight depending on the distance to the interpolation point. Recommended values are from 1 to 3.
8. Interp d – smoothing parameter. Recommended values are > 0.
9. X Min, X Max, Y Min, Y Max – map limits, in meters.

Fig. 106 illustrates an example of building a map without interpolation and with the "Show Data Points" flag enabled.

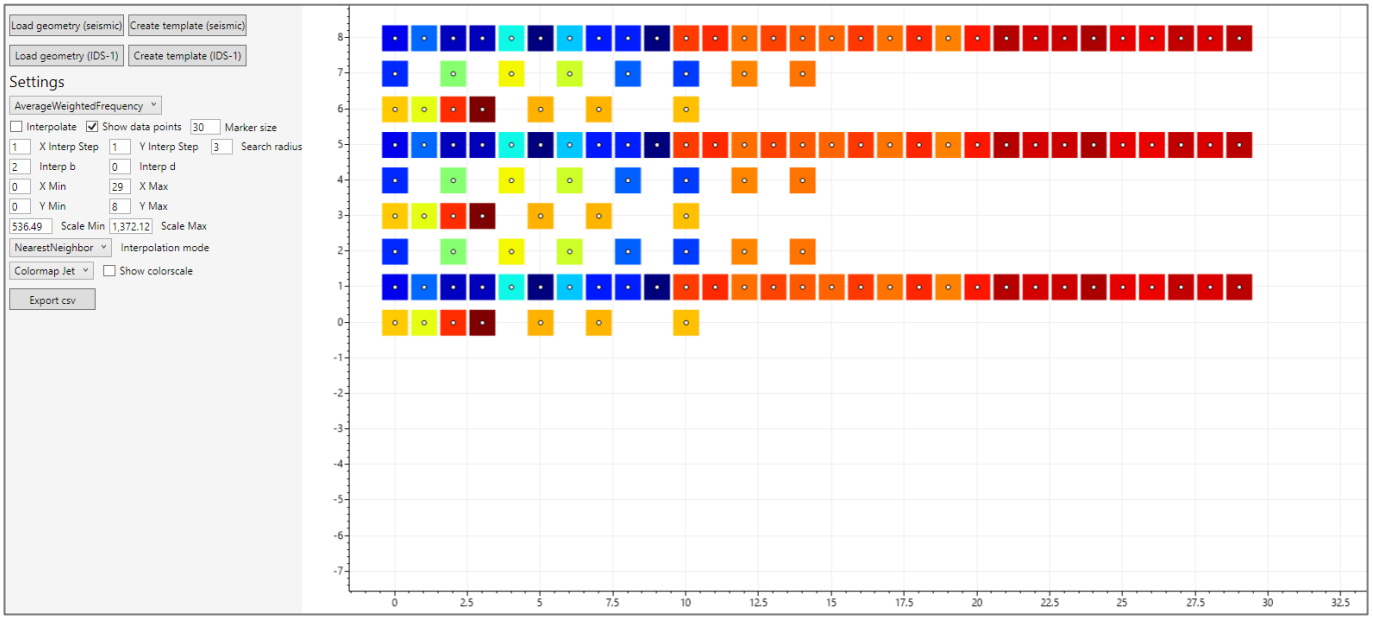

Fig. 106. Non-interpolated map with marked data points

Fig. 107 shows an example of building a map with interpolation and with the "Show Data Points" flag enabled.

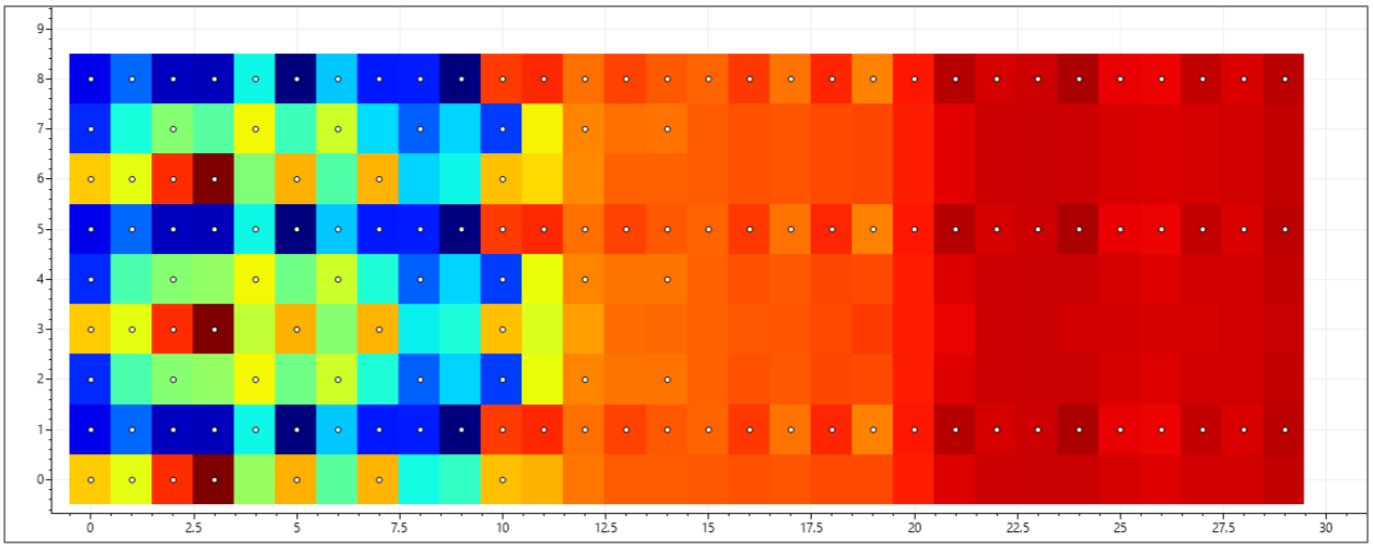

Fig. 107. Interpolated map with marked data points

To change the interpolation method, select an item from the list (Fig. 108).

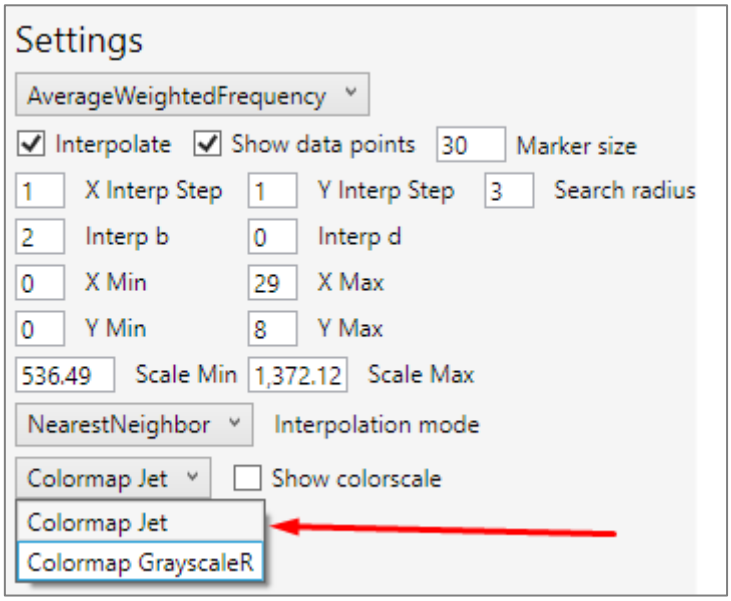

Fig. 108. Changing the interpolation method

Fig. 109 shows an example of building a map with a changed interpolation mode and a changed color scale.

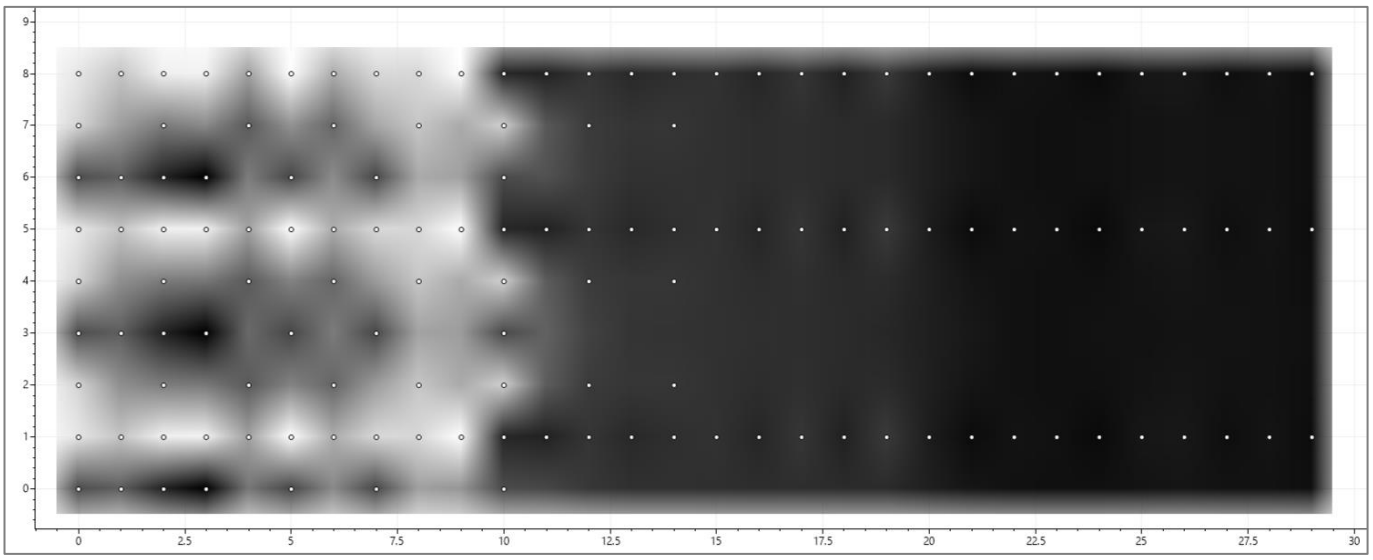

Fig. 109. Interpolated map with modified color scale and modified interpolation mode

5.4 Map export

To upload a table of data points with attribute values, click the "Export csv" button. In the save file dialog box, enter the file name. As a result of this operation the data points will be uploaded together with their coordinates to the csv table. The generated file can be used for building 2D and 3D maps/images in third-party software.

GEODEVICE

+7(812) 748-18-82
office@geodevice.ru
www.geodevice.ru