DECO-Geophysical Co. Ltd.

www.radexpro.ru

# *RadExplorer 1.4*

## The software for GPR data processing and interpretation

**User Manual**

World-wide distribution:

www.malags.com

**Moscow-2005**

# TABLE OF CONTENTS

# I. Introduction

**RadExplorer** is designed for GPR survey data processing and interpretation. The programme is compatible with formats of principle GPR producers, including: RAMAC/GPR (Mala Geoscience), SIR (GSSI), PulseEKKO (Sensors&Software), Zond (Radar Systems), OKO (Logis), and with SEG-Y international format with possibility of field redefinition.

The programme allows accomplishing the whole process of GPR survey data processing and interpretation within the framework of one single system: reading and visualization of radargram, DC removal, background removal, trace edit and spatial interpolation on equal profile interval, amplitude correction, deconvolution, 2D and bandpass filtering, migration, topography correction, determination of dielectric constant/electromagnetic wave propagation velocity, reflection picking, model construction, time to depth conversion.

Most of the processing routines determine their default parameters according to the record properties. If the user is unsure about selection of processing parameters, use of the default parameters may often ensure an acceptable result.

User-friendly interface allows quick and easy mastering the skills of working with the programme.

## Contacts

*DECO Geophysical Co. Inc.*
MSU Science Park, estate 1, building 77, office 104
Leninskie Gory, 119992 Moscow, Russia
Tel./Fax (+7 095) 930 84 14
Web: www.radexpro.ru
E-mail: support@radexpro.ru

*MALÅ GeoScience AB (main office)*
Skolgatan 11
S-930 70 Malå
Sweden
Telephone +46 953 34550
Facsimile +46 953 34567
Web: www.radexpro.ru
E-mail: support@malags.se

## How to use this manual

This manual is divided into several major sections:

Section **II. Installation** provides brief instructions how to install the software to your computer.

Section **III. Quick start** is the essential one and we recommend that everyone reeds it. It will guide you briefly through the main steps of working with the software from opening a file with raw GPR data until saving and printing out the results of data processing and interpretation. This section will provide you with enough familiarity to immediately start working with the programme. Whenever, during your work, you will need more information, you can find it in two other sections of this manual.

Section **IV. Input/Output routines** contains detailed description of the programme facilities for data input and output.

Section **V. Programme reference** contains detailed description on every feature of the programme. Menu commands, tool bar commands, visualization window, all the panels are discussed one by one and the related functionality is explained.

Some additional information is provided in the **Appendix**.

# II. Installation

To install the programme launch the Setup.exe file from **RadExplorer** installation disk and follow the instructions of the installation wizard. If you are installing the programme for the first time you will probably have to reset the computer.

*ATTENTION:* When installing the programme on a computer working under Windows NT/2000/XP operation system one should have the administrative rights.

# III. Quick start

Before you start working with this programme, be sure that the HASP-dongle is connected to the appropriate port of the computer (LPT or USB depending on dongle type).

You can launch **RadExplorer** through the "**Start"** menu on the Windows desktop. One can also launch the application through "**Explorer"**, **"My Computer"** or any other file system navigator. To do this, launch the file **Launcher.exe** from the folder where the programme has been installed.

## *Data input*

After the application has been launched for the first time, the main window and the dialog box offering you a selection of data input formats will be displayed (see Fig. 3.1).

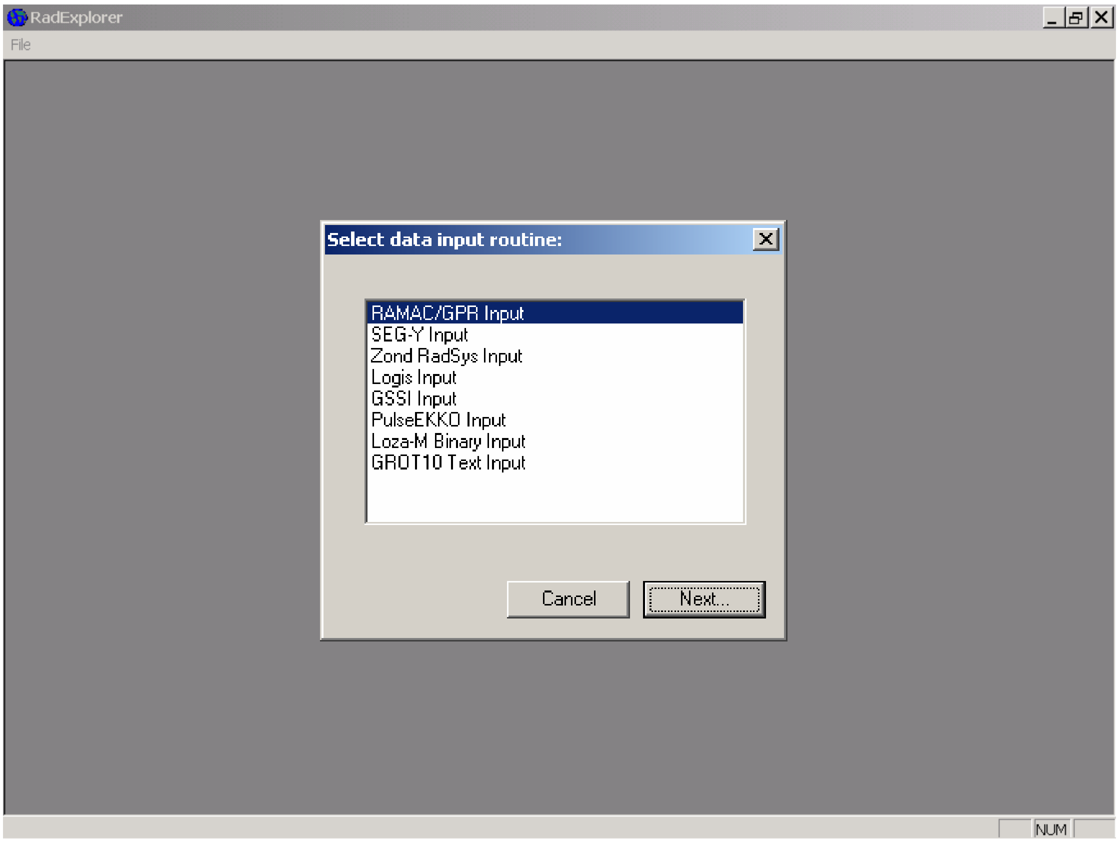



Fig. 3.1

Select the format you are interested in and press "**Next …**". The dialog box for the data loading routine will open. The dialog box appearance depends on the format you have selected (see "**Input/Output Routines**" section). For example, if you select LOGIS format, the dialog box represented on Fig.3.2 will appear.

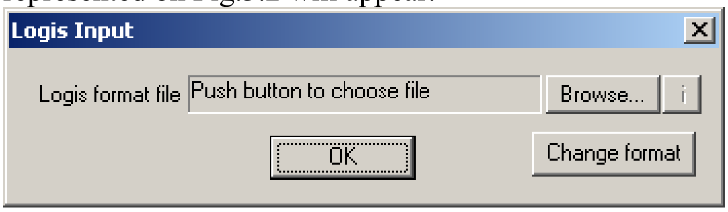



Fig. 3.2

Press the "**Browse …**" button and specify the data file. The file name will display in the "**File**" field. To confirm your choice press **"OK"**.

Next time you launch the application, you will immediately see the dialog box of the last input routine used. To change the input format, press **Change format** button.

## *First acquaintance with the software*

When a data file is loaded, the main working area of the programme will be displayed (see Fig. 3.3). It is divided into several functional sections:

- Menu;
- Tool bar;
- Visualization window;
- Processing routines, flow and model panel;
- Visualization and processing parameters panel.

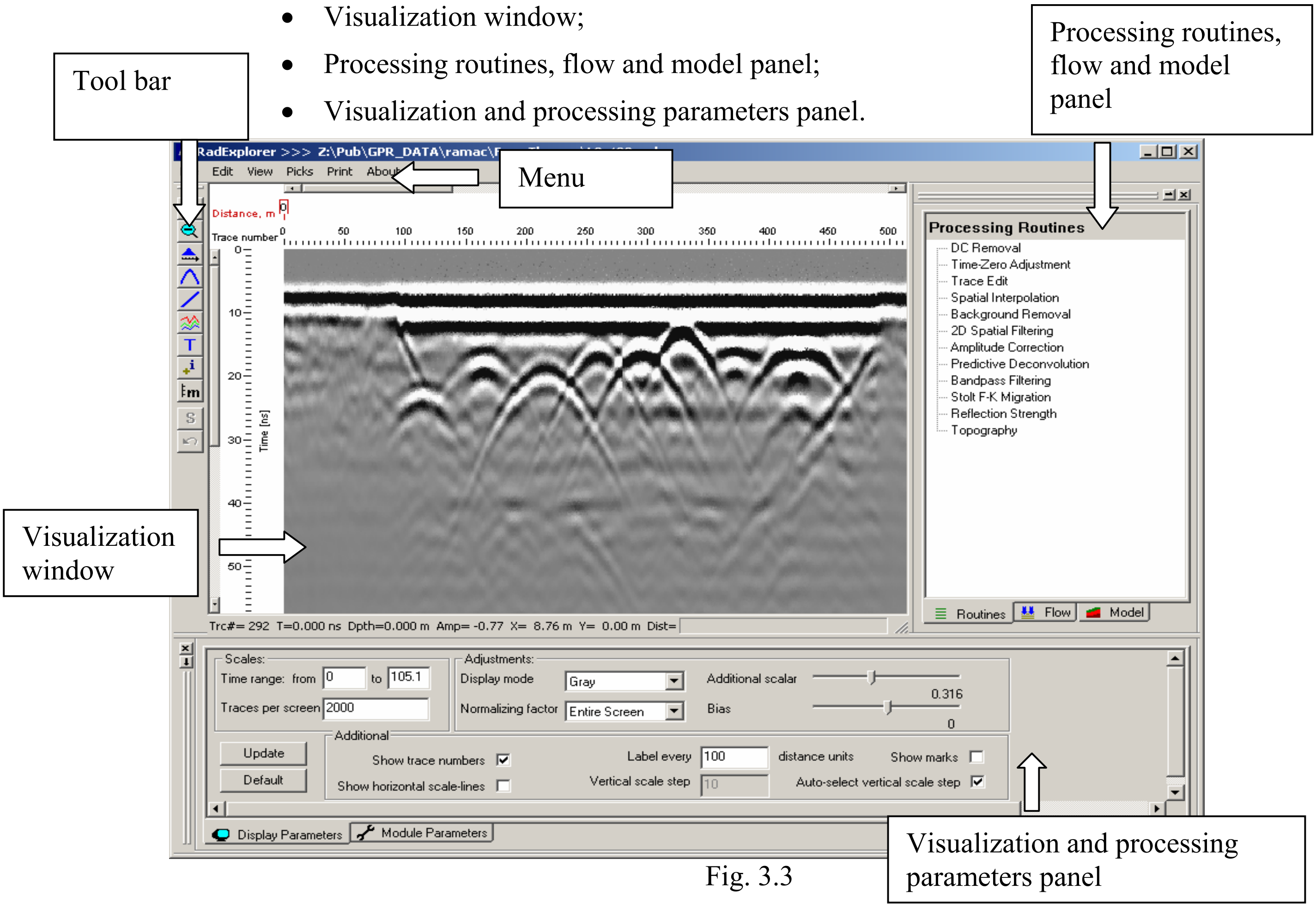



Fig. 3.3

Select “**Edit/Preferences…**” command of the menu to set up the desired distance units (meters or feet). By default, all the distances are measured in meters.

Visualization and processing parameters panel includes two tabs. The first one is available all the time and allows adjusting the parameters of data displaying on the screen: scale, display modes, gain factor (additional scalar), etc. The second tab is meant for processing parameters adjustment and its appearance depends on selected processing routine (in the model editing mode the model editor interface is also displayed there).

## *Data processing*

The processing routines, flow and model panel contains three tabs. In the first one the list of available processing routines is shown. Click on any routine and its parameters will be displayed on the second tab of the parameters panel (see Fig. 3.4).

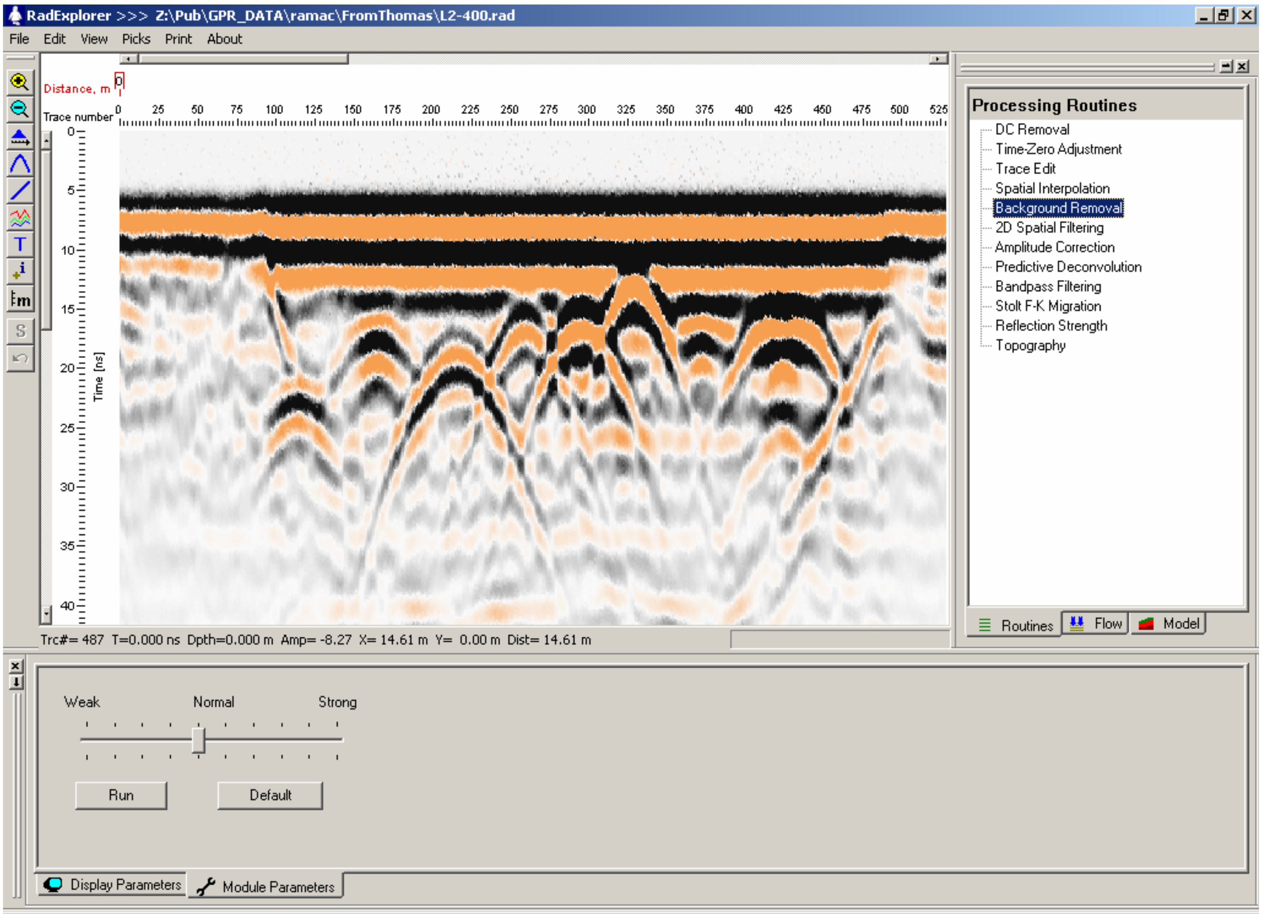



Fig. 3.4

To apply the routine, press "**Run**" button. The result of the processing will display on the screen. If you decide to change the routine parameters at this moment and apply the routine again, the result will be recalculated using the initial data but with new parameters. Thus, the repeated pressing of the "**Run**" button does not result in repeated application of the routine. When the routine is finished you can press the undo button  on the tool bar to undo the routine application result. The initial data that has been before the routine was applied will display.

If one press the  button on the tool bar after the routine has been applied, the result of the routine application will be saved in memory and the undo button  will become disabled (when the changes were saved in the memory one can always undo the routine through the "**Flow**" tab on the routine panel). Now if you press the "**Run**" button once again, the routine will be applied to *the already processed data*, i.e. it will be applied twice.

Most of the processing routines contain "**Default**" button. Use this button if you would like to roll back to the initial settings of the routine.

After you have gained the acceptable result of the routine application, select the next routine from the list. If you have not saved the result of the processing in memory, the window shown on Fig. 3.5 will display.

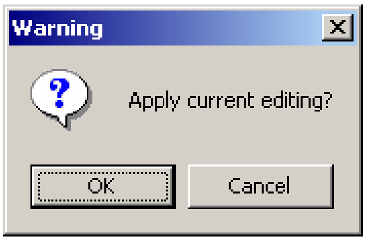



Fig. 3.5

If you press the "**OK**" button the result of previous routine application will be saved in memory. Use "**Cancel**" button to cancel the result of the previous routine. In either case, in the second tab of the parameter panel the parameters of newly selected processing routine will appear (see Fig. 3.6).

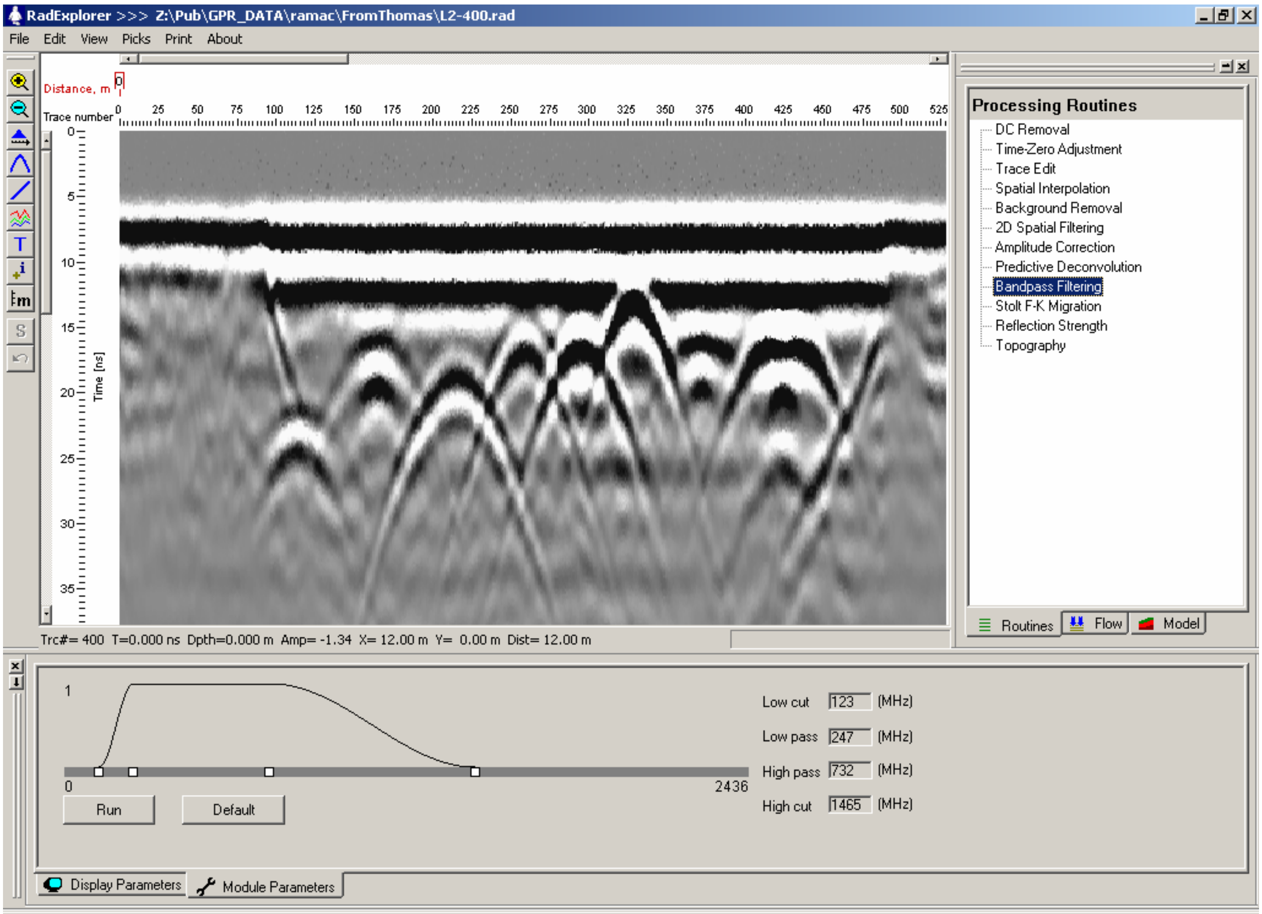



Fig. 3.6

*NOTE:* The routine application result will be also saved in memory (without request) when switching to the "**Flow**" or the "**Model**" tab of the routine panel.

All the processing routines applied to the data are automatically included into processing history or "flow". To work with the flow, you can use "**Flow**" tab (see Fig. 3.7).

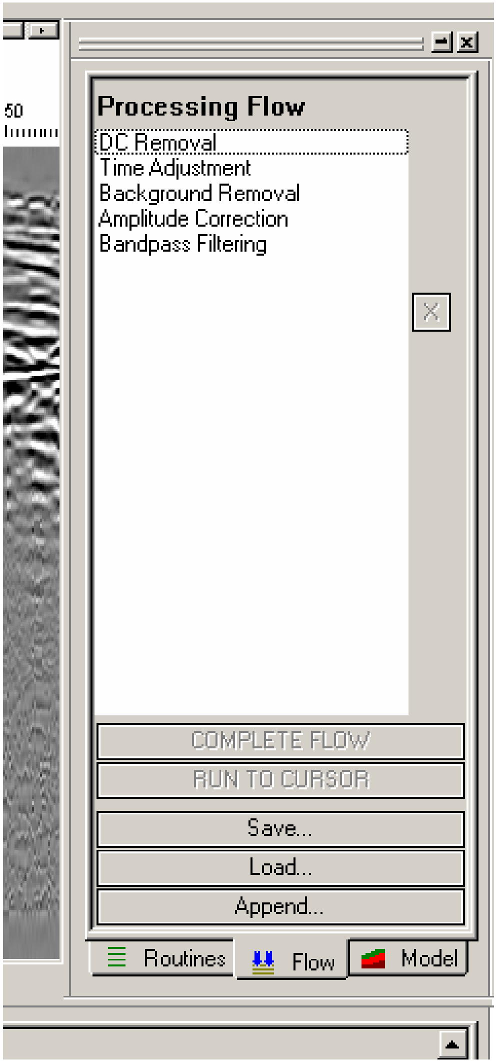



Fig. 3.7

When working with this tab you can undo the routines applied before, apply the undone routines again but with different parameters, delete the routines from the flow. The flows can be saved as files on hard disk, loaded from the disk and re-used with other files.

The flow is filled in top-down, i.e. the first applied processing routine will be displayed at the top of the flow, and the last routine that has been applied will be displayed at the bottom.

Click the left mouse button on the routine name in the processing flow. This routine and all the following processing routines will be undone. At the same time, in the second tab of the parameter panel, the parameters of the selected routine will appear for editing. To apply the whole flow with new parameters press “**Complete flow**”**.** To apply a part of the flow select with the mouse the last routine you would like to apply and press the “**Run to cursor**” button. All the routines, including the selected one, will be applied.

In the flow all the canceled (not applied) routines are being marked with an arrow. (see Fig. 3.8).

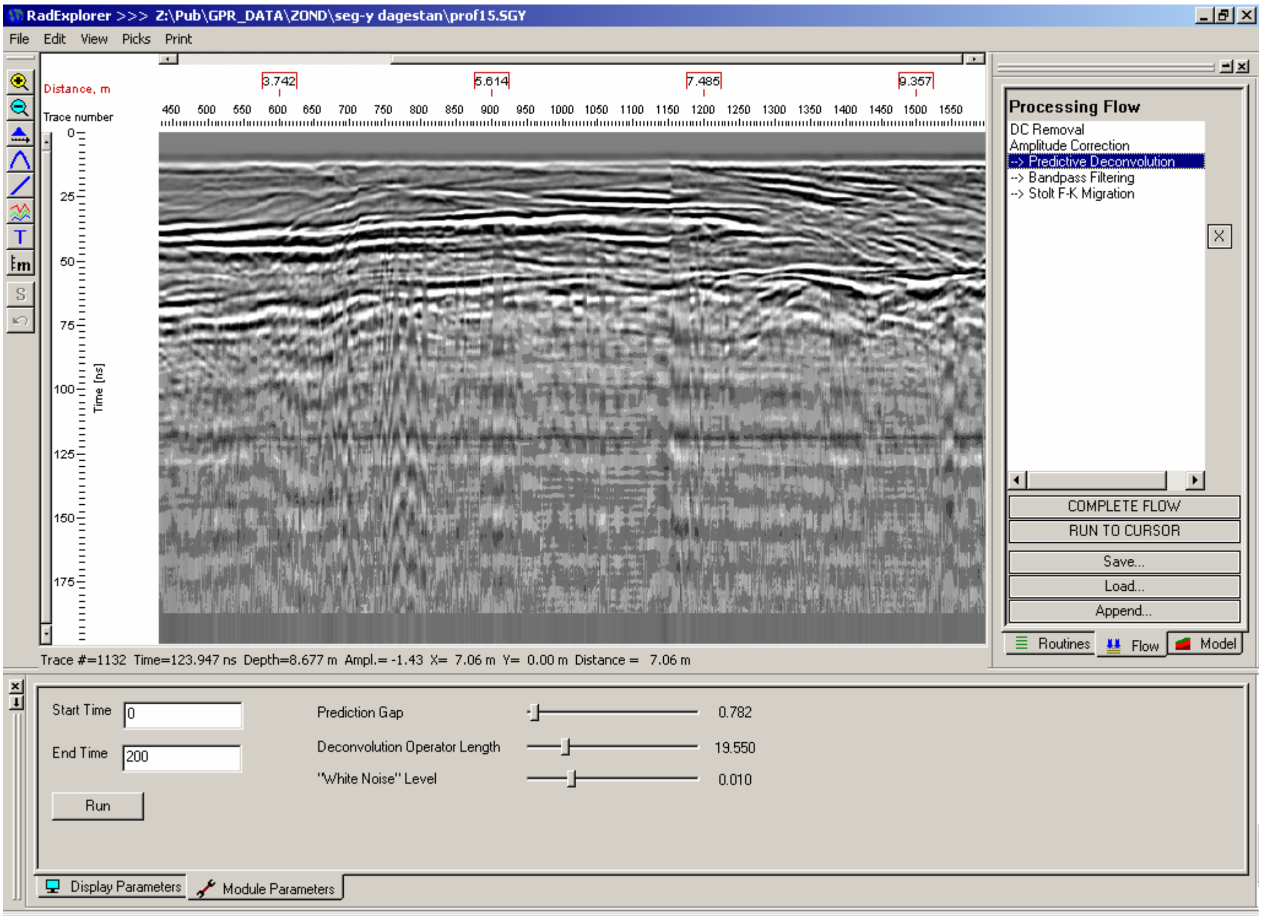



Fig. 3.8

On this Figure the *DC Removal* and *Amplitude Correction* routines have been applied. Besides, in the flow the *Predictive Deconvolution, Bandpass Filtering* and *Stolt F-K Migration* routines are in the canceled (not applied) state. On the second tab of the parameter panel, the parameters of the selected routine in the flow – *Bandpass Filtering* – which can be edited are displayed.

## Interpretation

When data processing is finished you can move to interpretation.

*IMPORTANT:* Though the software allows alternating data processing with interpretation, we strongly recommend that interpretation starts only after the processing is completed. Use/undo of some processing routines (e.g. *Topography* or *Time-Zero Adjustment*) with existing interpretation may result in moving of the interpretation elements relative to the data or/and in their significant distortion.

In **RadExplorer** programme the primary tool for data interpretation is the Velocity Model Editor.

Creation of an adequate model (velocity model) of the subsurface is the main and crucially important part of the interpretation. The model is primarily used for conversion of the radargram from two-way travel time scale to depth scale. However, beside this, virtually all depth values in this software are calculated with use of the velocity model. The depths appearing in the status bar, in the ‘info-marks’, at the ‘floating depth-ruler’, depths of picked horizons when saved into table text files – all these values are calculated basing on current velocity model! Thus, even if you prefer to work with so-called ‘layer interpretation’ picking reflections and saving them into table files, to ensure proper horizon depths it is crucially important to create an adequate velocity model before the horizon picks are saved.

The access to the model editor is accomplished through the “**Model**” tab of routine panel (see Fig. 3.9).

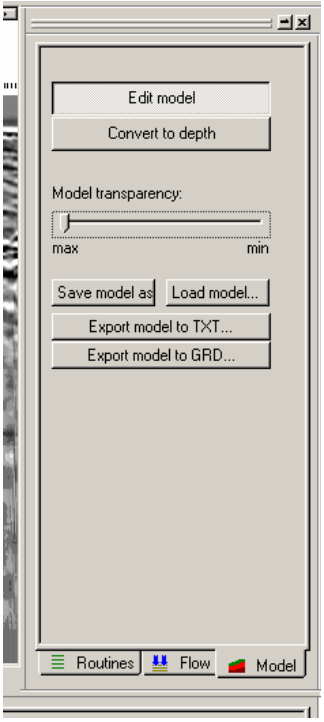



Fig. 3.9.

When moving to this tab, the model editor interface is displayed on the parameter panel (see Fig. 3.10).

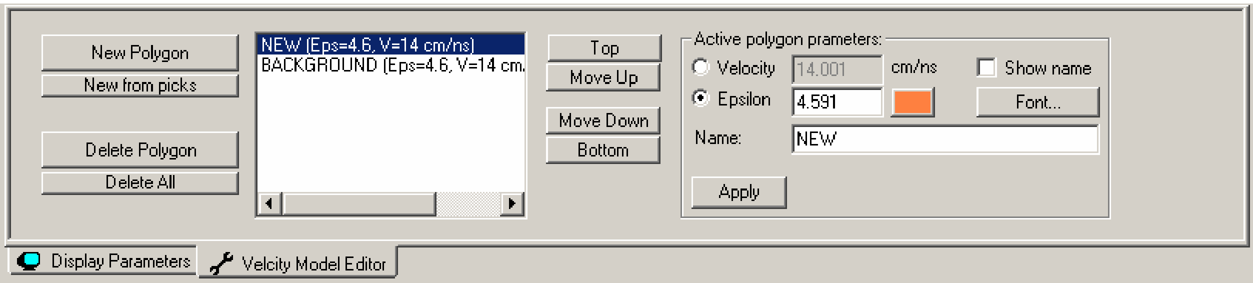



Fig. 3.10

In **RadExplorer** programme the model is a set of polygons with defined values of dielectric constant (electromagnetic wave propagation velocity). When opening a data file, the model containing only one polygon (BACKGROUND) is created by default. The background occupies the whole data area and is colored in gray by default, its dielectric constant is set as equal to that specified by GPR operator while recording (if this kind of information does not exist the value is set by default).

The application allows viewing the model and the radargram simultaneously by means of adjusting the transparency degree of the model with the “**Model transparency”** slider on the “**Model”** tab (see Fig. 3.9). At that, at the highest transparency only the radargram will show, and at the lowest transparency – only the model. Before you start working with the model editor, move the “**Model transparency”** slider in order to achieve the transparency degree for the model that suits you best.

To add a new polygon one should press the “**New polygon”** button in the model editor window (see Fig. 3.10). The line “**NEW**” corresponding to the newly created polygon will appear in the list of polygons. Specify the color and other polygon parameters. After that you can start drawing it on the screen by adding the dots by single-clicking the mouse button. The polygon right away becomes closed (see Fig. 3.11).

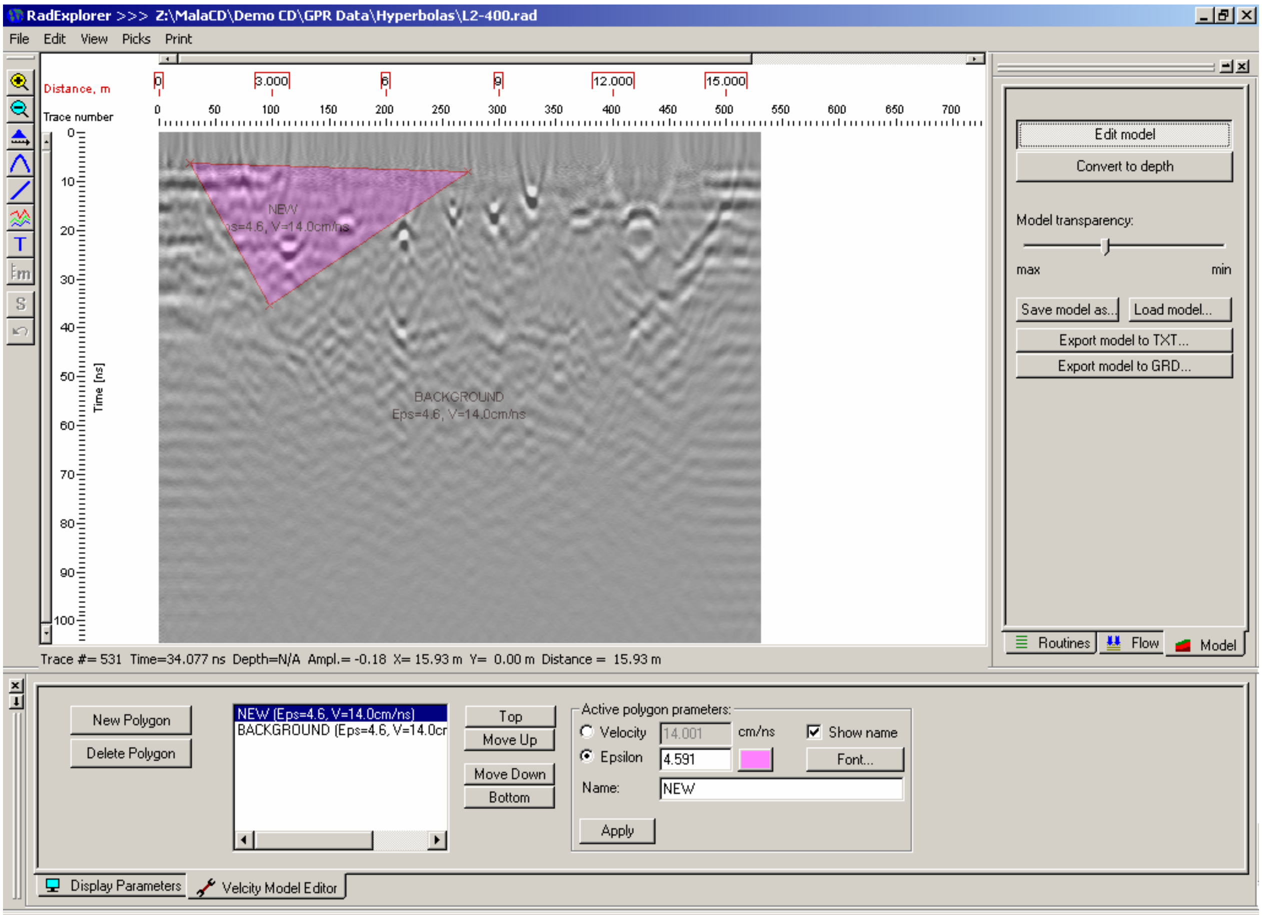



Fig. 3.11

You can move the already created dot by capturing it by the additional (right) mouse button and then dragging to the required position. To delete the dot double-click the additional (right) mouse button on it.

You can create as many polygons as you need. You can change the polygon position regarding one another at any time with the help of the “**Top”, “Move up”, “Move down”,**

**“Bottom”** buttons (see Fig. 3.12).

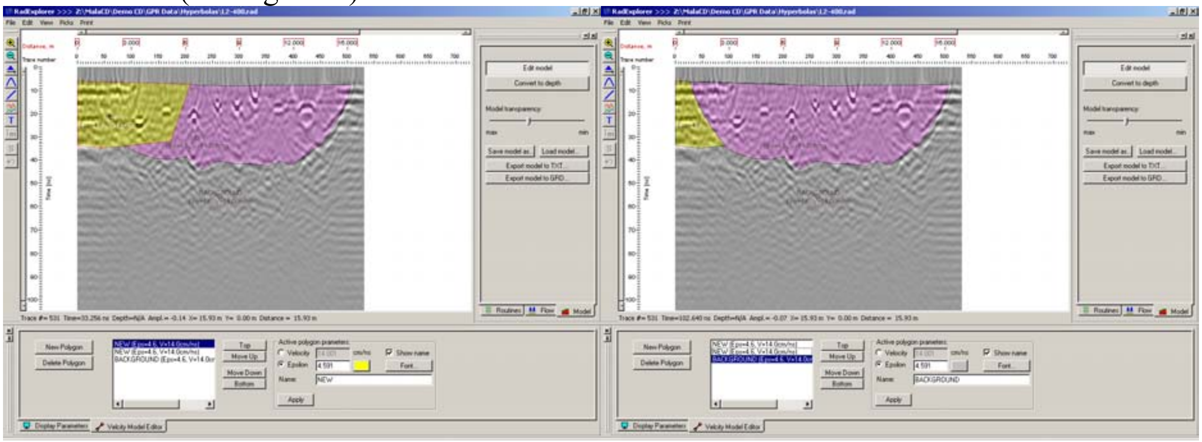


Fig. 3.12 – on the left a yellow polygon is placed on top of the pink one, on the right– vice versa.

The model can be used for radargram conversion from time scale to depth scale. To do this, one should push the **”Convert to depth”** button on the “**Model”** tab. A result of such conversion is shown on Fig. 3.13.

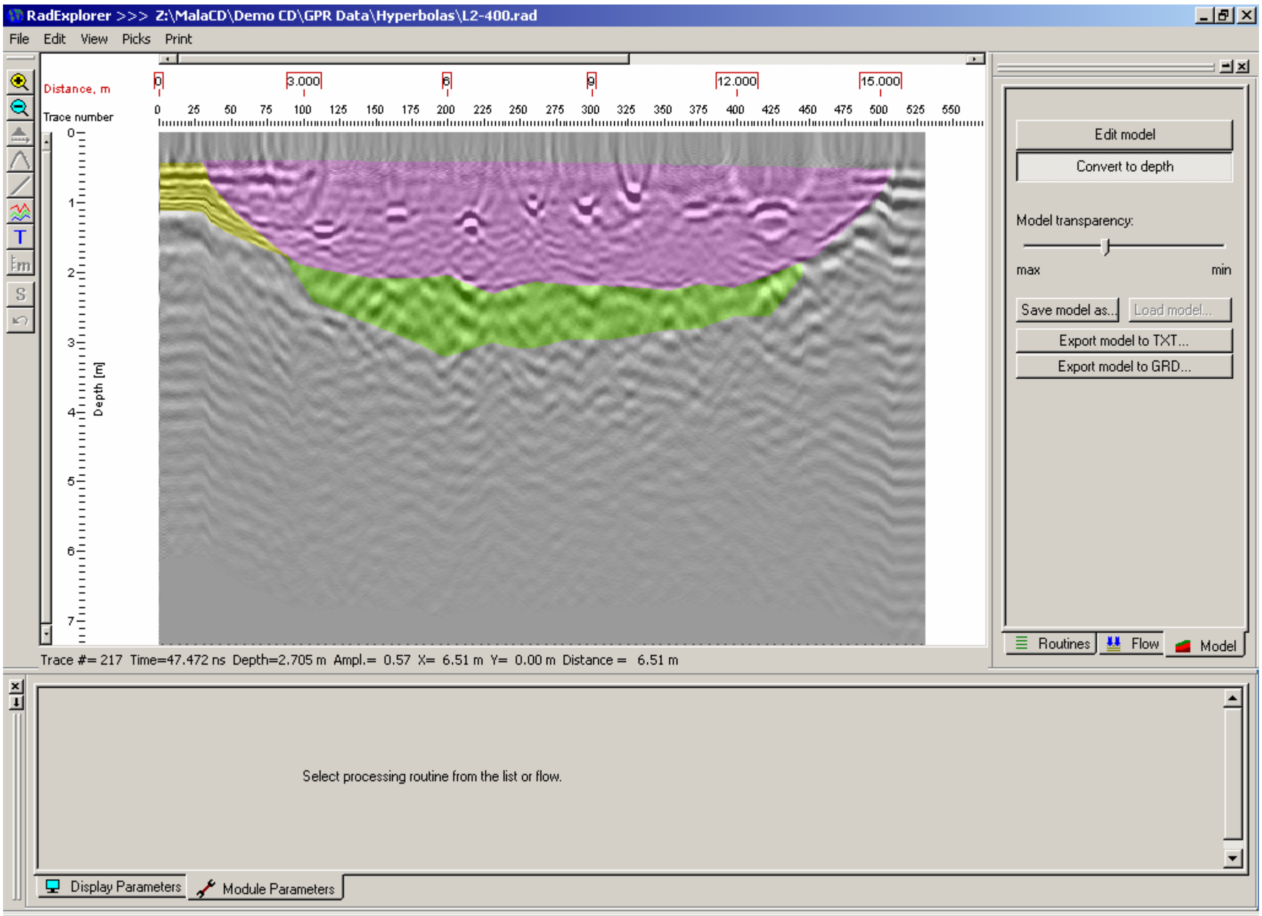



Fig. 3.13

The data in depth scale cannot be processed, that is why when switching from the "**Routine"** tab to the "**Flow"** tab the radargram is being automatically calculated back to the time scale. You can print out the data in depth scale or export them as a bitmapped image ("**Print/Print**…" or "**Print/ Save the image…"**). In addition you can export the model either as tabular text file or as a file with GRD extension – the standard grid of the Surfer programme.

In the **RadExplorer** programme, data interpretation tools, in addition to the model editor, comprise horizon picking, 'info-marks' and text marks. It is worth mentioning, that there is a possibility to convert 2 horizon picks into a polygon of the model. See corresponding sections of this manual for more details.

### *Saving results*

The processed data can be printed out, exported as a bitmapped image or saved (in time-scale) into a file of SEG-Y or RAMAC format ("**File/Save as…**" menu). At the same time the created model as well as picks and text marks will be saved into additional service files.

If the output file name is *myfile.sgy* than the velocity model will be saved into *myfile.sgy.vm* file, horizon picks, info-marks and text marks will be saved into *myfile.sgy.vo* file. These are the programme service files recorded in its internal format. When you re-open the saved data file, the model, picks, info-marks and text marks will be restored from these files in case they are located in the same folder with the data file.

By default, it is not allowed to save data file when it is converted to depth ("**File/Save as...**" command becomes disabled). This limitation was made because none of the GPR data formats used by **RadExplorer** support depth scale. However, in most cases, you will never need to save data in depth scale. Since, when the data is saved in the time scale its velocity model is saved as well, you can always re-open the fully processed data file together with its model and convert it to depth again with a single click on the "**Convert to depth**" button.

Interpretation elements (velocity model, horizon picks, info-marks) can be exported into text (ASCI) files. Beside, the model can be exported to GRD-files that can be easily opened by Surfer (Golden Software Inc.). You can learn how to export any particular element of the interpretation in the corresponding section of this manual. The export text formats of the interpretation elements are described in Appendix.

## IV. Input/Output routines

### *IV.1. Data input*

When the programme is launched for the first time the dialog box (see Fig. 4.1.1) offering selection of required format for data input will be displayed.

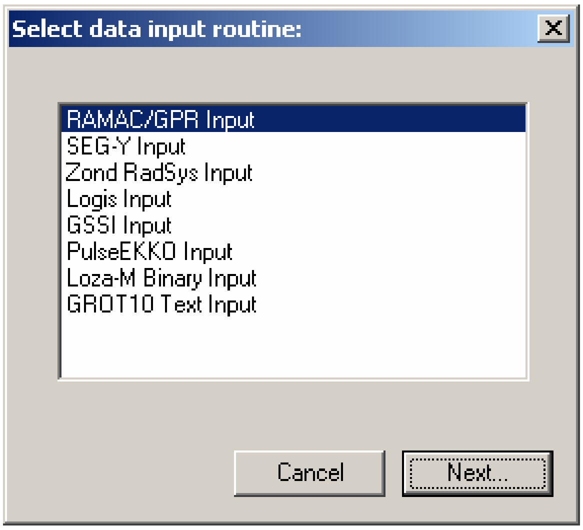



Fig. 4.1.1

Select the required format and press "**Next**". The window with parameters of selected data input procedure will open.

Next time the application is launched or "**File/Open**" menu command is selected you will see directly the dialog box of the last used data input format. To select another format, press **Change format** button.

## RAMAC/GPR

When selecting the RAMAC/GPR format data input routine the dialog box shown on Fig. 4.1.2 will appear.

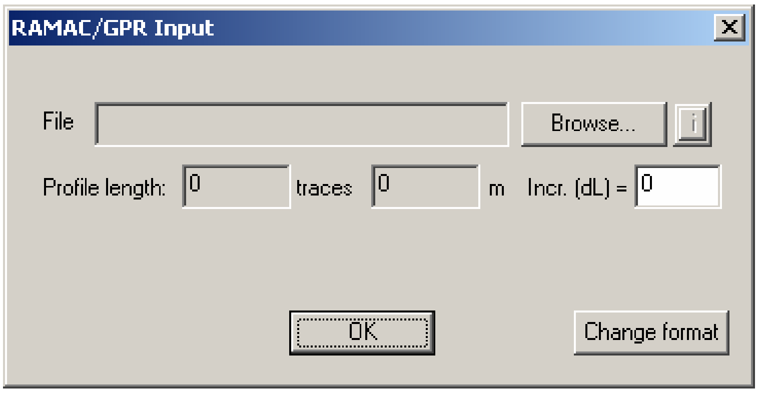



Fig. 4.1.2

Press the "**Browse**" button to select a file in RAMAC/GPR (*.rad file extension). After that information about the file (see Fig. 4.1.3) will appear in the dialog box fields.

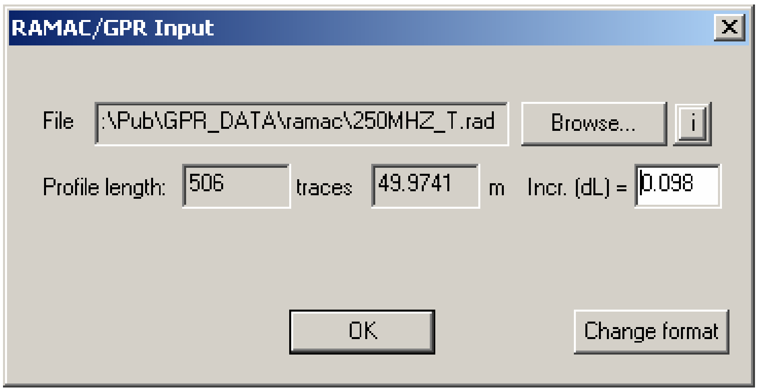



Fig. 4.1.3

In the "**Profile length**" group in the first field, the number of traces (GPR shots) recorded in the file will be displayed. In the second field, the profile length in meters will be shown, "**Incr. (dL)=**" will show the profile increment, i.e. distance in meters between the adjacent traces . In case if the profile increment is determined incorrectly you can change it by specifying the new value.

To confirm the data file selection press "**OK**". To select another input format press "**Change format**".

## SEG-Y

When selecting the SEG-Y format data input routine the dialog box shown on Fig. 4.1.4 will appear. Press the "**Browse**" button to select a file in SEG-Y format. The name of the selected file will show up in the "**File**" field.

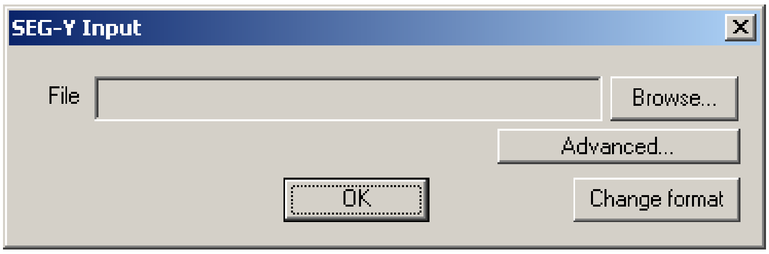



Fig. 4.1.4

To confirm the data file selection press "**OK**". To select another input format press "**Change format**".

When pressing the "**Advanced…**" button, the dialog box with information about the file and advanced parameters of the format will open (see Fig. 4.1.5). It is not recommended to change the advanced parameters without need. You should change the parameters only in case when the file in SEG-Y format is being read incorrectly.

*NOTE*: The more detailed description of SEG-Y format can be viewed on the site of the *Society of Exploration Geophysicists (SEG)* in *"Technical Standards"* section- http://seg.org/publications/tech-stand/

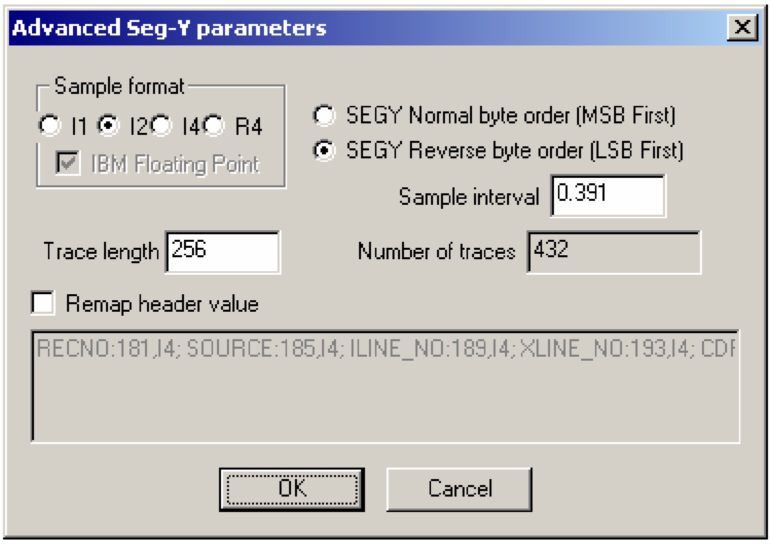



Fig. 4.1.5

In the “**Sample format** ” field select the number format used for data samples recording: **I1** – 1 byte integer number, **I2** – 2 byte integer number, **I4** – 4 byte integer number, **R4** – 4 byte floating point real number.

When selecting the “**R4”** option the “**IBM Floating Point**” option becomes available. If it is on the floating point numbers are supposed to be recorded in IBM format, otherwise – in IEEE format.

The “**SEGY Normal byte order (MSB First)**” or “**SEGY Reverse byte order (LSB First)**” options allow selection of byte order in number presentation.

The “**Sample interval**” field defines the distance between the trace samples expressed in nanoseconds. In case if the sample interval is determined incorrectly one can change it by specifying a new value in this field.

In the “**Trace length**” field the number of trace samples should be specified. In case if this value is determined incorrectly you can change it as well.

In the “**Number of traces**” field the number of traces in a file should be specified. The “**Remap header values**” option allows the user to specify in an explicit form from which byte of SEG-Y trace header and in which number presentation format should this or another information be read. The syntax of field redefinition is the following:

*HEADER_NAME: <initial_byte>, <number_format>*

The byte numbering starts with zero, the number format can be **I1** – 1 byte integer number, **I2** – 2 byte integer number, **I4** – 4 byte integer number, **R4** – 4 byte real number with floating point, **R8** – 8 byte real number with floating point.

It might be reasonable to remap, first of all, the following headers:

*REC_X* – X-coordinate of the trace
*REC_Y* – Y-coordinate of the trace
*Eps* – the dielectric constant value specified by the operator

To confirm the specified parameters press “**OK**”. Press “**Cancel**” to cancel the changes. When the window with extra parameters closes, press “**OK**” in order to confirm the data file selection and the other changes. To return to the data file format selection press “**Cancel**”.

## Zond (RadSys)

When selecting the Zond format (Radar Systems) data input routine the dialog box shown on Fig. 4.1.6 will appear. Push the “**Browse”** button to select the file in Zond RadSys format. The name of the selected file will show up in the “**File**” field. To confirm the data file selection press “**OK**”. To select another input format press “**Change format**”.


Fig. 4.1.6

## LOGIS

When selecting the LOGIS (OKO) format data input routine the dialog box shown on Fig. 4.1.7 will appear. Press the “**Browse**” button to select the file in LOGIS format. If you press the “**i”** button the window containing information about the selected file will open. To confirm the data file selection press “**OK**”. To select another input format press “**Change format**”.


Fig. 4.1.7

## GSSI

When you chose the GSSI format of data input the window shown on Fig. 4.1.8 will open. Press the “**Browse**” button to select the file in SIR GSSI (*.dzt file extension) format. The name of selected file will display in “**File**” field. To confirm the data file selection press “**OK**”. To select another input format press “**Change format**”.

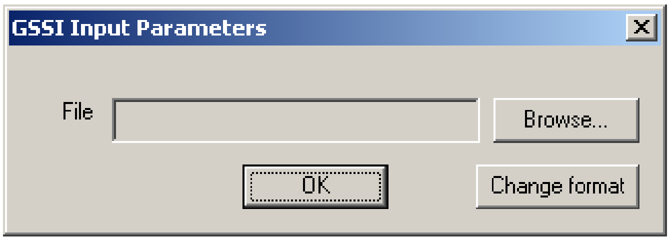



Fig. 4.1.8

### PulseEKKO

When you chose the PulseEKKO (Sensors&Software) format of data input the window shown on Fig. 4.1.9 will open. Press the “**Browse**” button to select the file in PulseEKKO (*.dt1 file extension) format. The name of selected file will display in “**File**” field. To confirm the data file selection press “**OK**”. To select another input format press “**Change format**”.

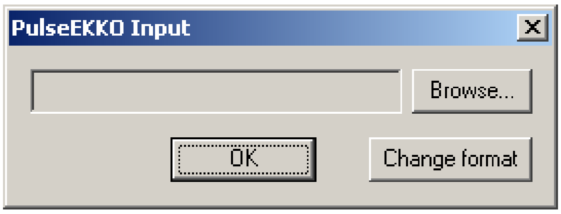



Fig. 4.1.9

## *IV.2. Data output*

By default, “**File/Save as…**” menu command is enabled only when the data is in time scale. After converting radargram to depth, the command becomes disabled not allowing you to save data. This limitation was made because none of the GPR data formats used by **RadExplorer** support depth scale. However, in most cases, you will never need to save data in depth scale. Since, when the data is saved in the time scale its velocity model is saved as well, you can always re-open the fully processed data file together with its model and convert it to depth again with a single click on the “**Convert to depth**” button.

*HINT:* Sometime there might be a need to save data in depth scale for the purposes of data exchange with other software. In this case, if you really need this you can do this. In the main menu select “**Edit /Preferences**” command and in the appearing dialog box set check the “**Allow saving in depth scale**” check-box. Then, after converting the data to depth the “**File/Save as...**” command will not get disabled. However, one shall be always aware of the fact that a data file saved in depth scale *cannot* be re-opened correctly by **RadExplorer**. All interpretation, including the model, will be lost as well. We strongly do not recommend saving data in depth scale unless for the purposes of data exchange. When you check the “**Allow saving in depth scale**” check-box you will see an appropriate warning message.

When the “**File/Save as…**” menu command is selected for the first time, the dialog box that offers to specify the data file format you are interested in will display (see Fig. 4.2.1).

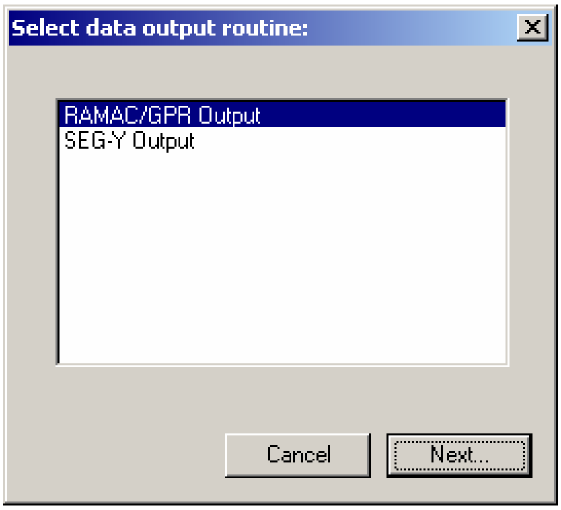



Fig. 4.2.1

At present, two output formats are supported by the programme – RAMAC/GPR and SEG-Y. We are strongly advising you to use the SEG-Y format everywhere it is possible because it is a an open internationally used format, which is understood by most of the GPR survey and seismic data interpretation programmes.

With the help of the mouse select the output data format you are interested in and press the “**Next**” button. The window with selected data output procedure parameters would open.

Next time the “**File/Save as…**” menu command is selected, you will immediately see the dialog box of the last used data output routine.

## SEG-Y

When selecting the SEG-Y format data output routine, the dialog box shown on Fig. 4.2.2 will open. Press the “**Browse**” button and, in the standard dialog box, select already existing or specify a new name for the file you would like to save the data in. The name of the selected file will display in “**SEG-Y File”** field.

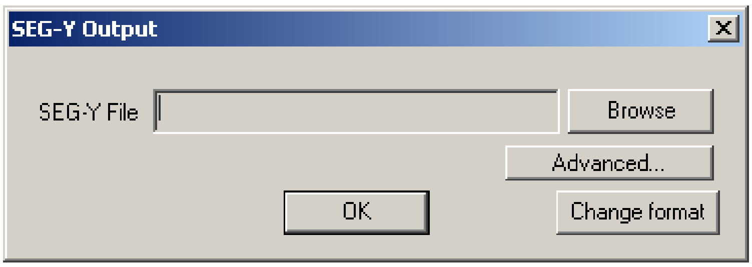



Fig. 4.2.2

Press “**Yes**” to confirm the file selection. To select another output format press “**Change format**”.

When pressing the “**Advanced…**” button the window that allows adjusting the advanced format parameters will open (see Fig. 4.2.3). We are strongly advising you not to change the extra parameters without need.

*NOTE*: The more detailed description of SEG-Y format can be viewed on the site of *Society of Exploration Geophysicists (SEG)* in *"Technical Standards"* section- http://seg.org/publications/tech-stand/

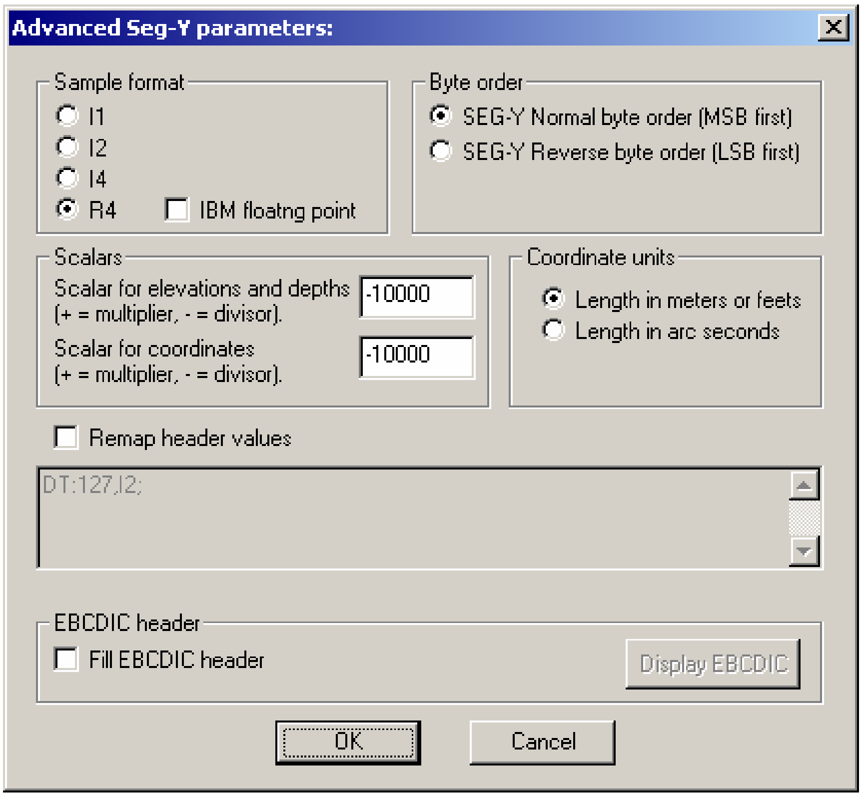



Fig. 4.2.3

In the "**Sample format**" field select the number format used for data samples recording: **I1** – 1 byte integer number, **I2** – 2 byte integer number, **I4** – 4 byte integer number, **R4** – 4 byte real number with floating point.

When selecting the "**R4**" option the "**IBM Floating Point**" option becomes available. If it is on, the numbers with floating point are recorded in IBM format, otherwise – in IEEE format.

In the "**Byte order**" field the "**SEGY Normal byte order (Senior First)**" and the "**SEGY Reverse byte order (Minor First)**" options allow selection of byte order in number presentation.

In the "**Scalars**" filed separately specify the values of scalars which will be used for saving the coordinates and excesses. The scalars are used for proper recording of values with floating point into integer format, and for their further correct decoding. Thus, when the trace coordinates are specified in meters the multiplier used by default – 10000 provides correct recording of coordinates with excess, in most of the cases, accuracy up to 0.1 mm. For example, the 1.54321 m value when recording into SEG-Y format will be multiplied by 10000 (=15432.1) and approximated to integer (=15432). On further reading of this number the programme having read the multiplier from the format, will divide the coordinate value (15432) by it and will restore the 1.5432 m value.

In the “**Coordinate units**” field one can select information on the type of coordinates that will be recorded into file format.

The “**Remap headers values**” option allows the user to specify in an explicit form in which byte of the SEG-Y trace header and in which number presentation format should this or another information be written. The syntax of field remapping is the following:

*HEADER_NAME: <initial_byte>, <number_format>*

The byte numbering starts with zero, the number format can be **I1** – 1 byte integer number, **I2** – 2 byte integer number, **I4** – 4 byte integer number, **R4** – 4 byte real number with floating point, **R8** – 8 byte real number with floating point.

It may be reasonable to remap, first of all, the following headers:

*DT* – sample interval
*NUMSMP* – the number of reports in each trace
*REC_X* – X-coordinate of the trace
*REC_Y* – Y-coordinate of the trace
*Eps* – the dielectric constant specified by the operator

When saving the data in SEG-Y format the programme fills the EBCDIC (the first 3200 bytes of SEG-Y file) header by zeros by default. If there is a need to fill in this part of the header, activate the “**Fill EBCDIC header**” in “**EBCDIC**” field. After that, you may click the “**Display EBDIC**” button to view and edit the EBCDIC header that will be recorded into file.

To confirm the set parameters press “**OK**”. Press “**Cancel**” to cancel the changes.
After the window with additional parameters is closed in order to confirm a file selection and all the changes of output parameters press “**OK**”. To return to the data format selection press “**Cancel**”.

## RAMAC/GPR

When selecting this procedure the dialog box shown on Fig.4.2.4 will open.

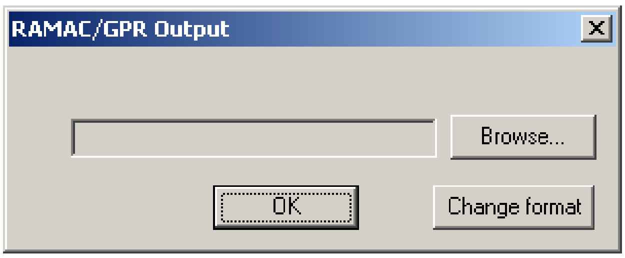



Fig. 4.2.4

Click the “**Browse…**” button and in a standard dialog box select already existing or enter a new name of a file where the data should be saved in. The selected file name will appear in the field to the left from the "**Browse…**" button. Click "**Yes**" to confirm a file selection. . To select another output format press "**Change format**".

# V. Programme reference

## *V.1. The menu*

The menu (see Fig. 5.1.1) gives access to the main functions that allow operations with data files, window view, tools for working with picks and printing parameters. The menu contains the following options:

- "**File**" – contains functions for operations with files;
- "**Edit**" – contains functions for the undoing the last processing routine, geometry definition, profile reversal, and programme preferences;
- "**View**" – contains functions for manipulations with working area;
- "**Picks**" – contains functions for operations with picks;
- "**Print**" – contains functions for printing and saving images.


Fig. 5.1.1

Let us consider the features of every menu.

### "File" menu

If a data file is opened all the submenus will be available, namely:

- "**Open**" – open a new file;
- "**Save as…**" – save the opened file in SEG-Y or RAMAC format;
- "**Close**" – close the opened file
- "**Exit**" – exit the programme.

By default, "**File/Save as…**" menu command is enabled only when the data is in time scale. After converting radargram to depth, the command becomes disabled not allowing you to save data. This limitation was made because none of the GPR data formats used by **RadExplorer** support depth scale. However, in most cases, you will never need to save data in depth scale. Since, when the data is saved in the time scale its velocity model is saved as well, you can always re-open the fully processed data file together with its model and convert it to depth again with a single click on the "**Convert to depth**" button.

*HINT:* Sometime there might be a need to save data in depth scale for the purposes of data exchange with other software. In this case, if you really need this you can do this. In the main menu select "**Edit /Preferences**" command and in the appearing dialog box set check the "**Allow saving in depth scale**" check-box. Then, after converting the data to depth the "**File/Save as...**" command will not get disabled. However, one shall be always aware of the fact that a data file saved in depth scale *cannot* be re-opened correctly by **RadExplorer**. All interpretation, including the model, will be *lost* as well. We strongly do not recommend saving data in depth scale unless for the purposes of data exchange. When you check the “**Allow saving in depth scale**” check-box you will see an appropriate warning message.

The “**Open**” and the “**Save as…”** menus are discussed in more detail in “**IV Input/Output Procedures**” section.

When selecting the “**Close**” or “**Exit**” submenus, if the changes in the opened file were not saved, the window with the warning message (see Fig. 5.1.2) will appear where it will be offered to save the current changes into the file with a new name.

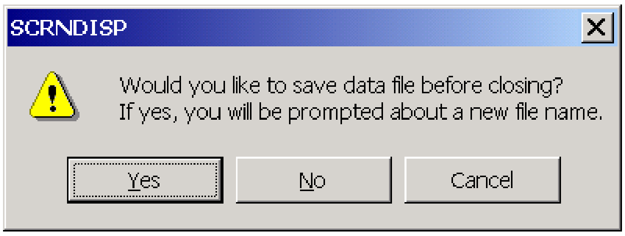



Fig. 5.1.2

In order to select the name of a file or to enter a new one click the “**Yes**” button. To close the file without saving the current changes press the “**No**” button, to return to work press the “**Cancel**” button.

## “Edit” menu

The “**Edit**” menu contains tree commands:

- “**Reverse profile**” command allows changing the direction of the profile display (from left to right or *vice versa*). This does not affect the distance values along the line – in either case the distances are calculated from the leftmost data trace. The command is available only for unprocessed radargrams. (If you have already applied some processing routines you will have to undo them with the help of the “**Flow**” tab on the routine panel, then reverse the profile and again accomplish the processing flow to the end). Note, that if any interpretation (velocity model, horizon picks, etc.) has been made, they will NOT be reversed. Thus, it is recommended that you first select the preferable direction of the profile display and then start making any interpretation.
- “**Define geometry**” command allows the user to define geometry of data file profile. The command is available only for unprocessed radargrams. (If you have already applied some processing routines you will have to undo them with the help of the “**Flow**” tab on the routine panel, then define the geometry and again accomplish the processing flow to the end).
- “**Preferences**” command shows a dialog box allowing set up of some general preferences of the software that are stored in registry. In particular, one can select here distance units (meters or feet) and allow saving data in depth scale (by default not allowed – see explanation for “**File/Save as…**” command).
- “**Copy screen**” command copies the current content of visualization window into the Windows clipboard (one can also use Ctrl+C for that).
- “**Undo one step**” command allows to undo the last processing step until the result is saved in

memory (this command is similar to the  button on the toolbar).

## Profile geometry definition

Profile geometry represents the information on data binding to the absolute or relative square coordinates on the site. Coordinates are specified in meters. To define the geometry 2 spatial coordinates – X and Y – are used. Comparing to UTM coordinates, X here is northing, Y is easting. (If only the distance from the profile origin that matters than the Y coordinate may not be specified by indicating 0 in respective fields). Based on these coordinates the programme calculates the distance between every trace and the profile origin. The correct binding is necessary for proper operation of such data analysis tools as “Hyperbola” and “Line” and for several processing routines (*Spatial Interpolation, Stolt F-K Migration*)

To define (redefine) profile geometry of the radargram, select the “**Define geometry**” command from the menu and specify the required parameters in the window that opens (see Fig. 5.1.3).

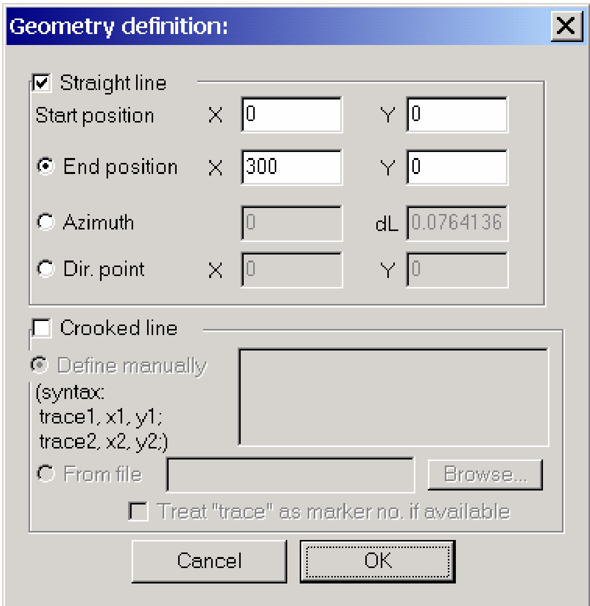



Fig. 5.1.3

Select the “**Straight line**” option in case if the selected profile is linear with known initial point and even profile interval. Specify the coordinates of the profile initial point in the “**Start position X__ Y__**” fields.

In this case, there are three ways to set the coordinates:

1. If the coordinates of the profile end are known one can activate the “**End position**” option and set X and Y coordinates in the corresponding fields;
2. If the geographic azimuth (according to compass) and the profile step are known one can activate the “**Azimuth**” option and specify the azimuth expressed in degrees in the corresponding field. Specify the profile interval in the “**dL**” field.
3. If the profile step and the coordinates of a point the profile went through are known one can activate the “**Dir. point”** option and specify its coordinates in the “**X**” and “**Y**” fields. Specify the profile step in the “**dL**” field.

If the profile under consideration is a broken line or the profile interval was changed or the coordinates of the starting point are unknown but the coordinates of some reference points are known, one can activate the “**Crooked line**” and set the coordinates of the points known in the profile. Here are two ways to chose:

1. To set the coordinates manually in the “**Define manually**” field according to the template *trace_number, X, Y; trace_number, X, Y; … trace_number, X, Y*;
2. To read from the tabular text file. To do this one should activate the “**From file**” option, click the “**Browse**” button and select the requited file in the dialog box that will open. The file must contain three space separated columns: the trace number – in the first column, X – in the second column, Y – in the third column:

   *trace_number X Y*
   *trace_number X Y*
   …
   *trace_number X Y*

If one activates the “**Treat “trace” as marker number**” option than in both cases of the crooked line geometry setting, the *trace_number* field value would be interpreted as a reference mark number that has been put by the operator (make sure that the mark you are using are presented in the file!).

Between the set points (and beyond them) values of coordinates will be interpolated linearly.

To confirm the geometry definition press “**Ok**”. If the parameters are specifies incorrectly the programme may show a warning message. Press “**Cancel**” in geometry definition field and it will close without saving the changes.

## “View” Menu

“View” menu gives the user opportunity to manage the programme’s working area. The two items are available in drop-down menu:

- “**Parameters**” – opens or closes visualization and processing parameters panel;
- “**Routines**, **Flow and Model**” – opens and closes processing routines, flow and model panel.

## “Picks” menu

This submenu contains commands for working with picks – horizon correlation lines. This submenu contains the following commands:

- “**New Pick …**” – create a new pick
- “**Delete Active Pick**” – delete the current pick
- “**Load Pick …**” – load the pick from the text file
- “**Save as …**” – save the current pick into a text file
- **“Export pick table…”** – export all picks into a single text table file
- “**Picking Parameters …**” – define the picking parameters
- “**Line Style …**” – determine the line style of the current pick
- “**Label picks …**” – adjusting the picks’ name displaying
- “**Next pick**” – make the next pick active (current)

- “**Previous pick**” – make the previous pick active (current)
- “**Name active pick**” – specify the name for the active pick

All the menus are active only in working with picks mode.

## Picking mode

To enter the *picking mode* press the  button on the toolbar. If there is no pick on the screen at this moment, a new pick will be created automatically. In the upper left corner of the screen the rectangular icon will display indicating that the programme is in *picking mode* (see Fig. 5.1.4).

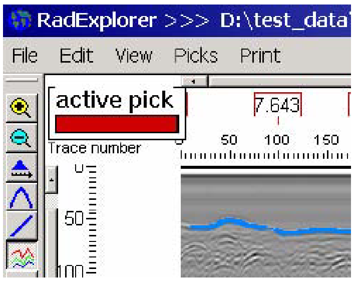


Fig. 5.1.4

The color of the icon indicates the color of the active (current) pick. When right-clicking this rectangle, the contextual menu for working with picks equal to the “**Picks**” submenu of the main menu of the programme will open. In case there are several picks on the screen, double-clicking the mouse button on the icon will make the next pick active (similar to the **Picks/Next** command of the menu).

When pressing the  button on the toolbar, if there is no pick on the screen the new pick will be created automatically. Now one can place the pick points on the radargram by simply clicking the main (left) mouse button.

***The main commands for pick editing:***

- *Add point* command is accomplished by clicking the left button of the mouse. The point will be displayed where the cursor is placed.
- *Relocate point* command is accomplished by the right button of the mouse. Capture the point that you would like to replace by the right button of the mouse and keeping it pressed drag the point to the requited place. You can also place the cursor to the place where you would like to relocate the point to and click the right mouse button. By doing this the point closest to the cursor position will relocate.
- *Delete point* command is accomplished by double-clicking the right (additional) mouse button. By doing this the point closest to the cursor position will be deleted.
- To *delete a group of points at once* press Shift key and push left mouse button at a corner of the area containing points to be deleted. Holding both the mouse-button and the key pressed, move cursor to the opposite corner of the area and release the mouse-button. If the area

selected contains points of the active pick, a message box will appear asking confirmation to delete the selected points.

- *Up-and-down displacing of the whole pick* is accomplished by dragging the pick by the right mouse button simultaneously holding the Shift key pressed.

In picking mode all the commands of the "**Picks**" menu become available.

"**New pick …**" – this command allows the user to create a new pick. The color of a new pick will display in the icon of working with picks mode. By repeating this command the user can create any number of picks.

"**Delete Active Pick**" – this command allows the user to delete active pick as whole.

"**Load Pick …**" – this command allows the user to load previously saved pick from the text file. When refer to this command the standard dialog box offering to select the file for picks will open.

**"Save as …"** – this command allows the user to save active pick into text file. When refer to this command the standard dialog box for data saving will open. Enter a new name or select the already existing file.

"**Export pick table…**" – when this command is selected, a dialog box with export parameters will appear. Indicate export interval and export value type (two-way times or depths according to the current model). After the parameters are set, press "**Next**" to select file name in a standard file dialog. The created text file will contain a table where for each interval along the profile the following values will be exported: X and Y coordinates, geographical coordinates (if available), times/depths of every pick. If at a certain interval some value is not available, "N/A" will be printed instead of the value.

"**Picking Parameters …**" – this command is meant for selection of mode and parameters of horizon correlation on the radargram. When refer to this command the dialog box shown on Fig. 5.1.5 will open.

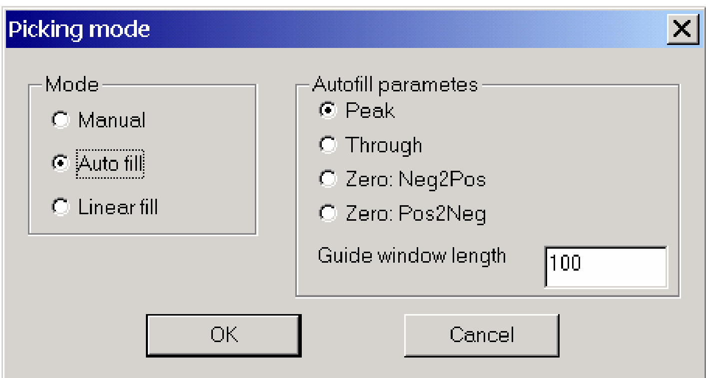



Fig. 5.1.5

In "**Mode**" field one can chose the mode of correlation:

- "**Manual**" – manual picking without interpolation. The points appear only where the user has placed them;

- “**Autofill**” – picking with horizon tracing. When the user adds a point, not only this point appears but also the points on all traces between this point and the previous one. The settings in the “**Autofill parameters**” field define the way the programme places the intermediary points;
- “**Linear fill**” – picking with linear fill between the adjacent points. When the user adds a point, not only this point appears but also the points on all the traces between the new picking point and the previous ones. Their position is defined by means of linear fill between the new picking point and the previous ones;

If the “**Autofill**” mode is chosen, the way the programme will try to trace the boundary will be defined in the “**Autofill parameters**” field:

- “**Peak** – picking will be accomplished according to local peak values of the signal:
- “**Through** - picking will be accomplished according to local minimum values of the signal (negative value with peak amplitude);
- “**Zero: Neg2Pos**” - picking will be accomplished according to the places of signal passage through zero from the negative values of the signal to the positive ones;
- “**Zero: Pos2Neg**” - picking will be accomplished according to the places of signal passage through zero from the positive values of the signal to the negative ones.

In the “**Guide window length**” field one can specify the window length (in nanoseconds) where selected parameter guiding (i.e. local maximum, minimum, etc.) will be accomplished.

To confirm parameter selection press “**Ok**”, otherwise press “**Cancel**”.
The “**Line style …**” command allows the user to change the type, thickness and color of the active pick. When this command is activated, a dialog window (see Fig. 5.1.6) will appear. There, in the “**Line type**” field one can select the line type for the pick and in the “**Line length**” one can specify the line thickness expressed in pixels.

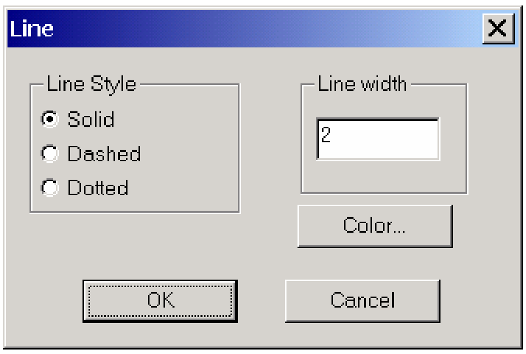



Fig. 5.1.6

When pressing the “**Color…**” button the standard dialog box for color selection of the pick will appear (see Fig. 5.1.7).


Fig. 5.1.7

When the “**Label picks …**” command is executed, the dialog window (Fig. 5.1.8) will open. There one can activate the pick labeling option (to do this one should switch on the “**Label picks**” option). The user can specify the “**Font size”** that will be used for signing the picks. Changing of these parameters will affect not only the current pick but also the rest of picks on the screen.

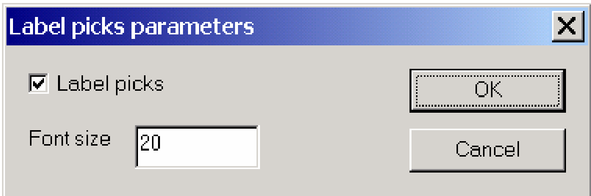



Fig. 5.1.8

After the “**Label picks**” option of the “**Label picks …**” menu has been activated the “*noname*” label will appear near the pick in the rectangular area (see Fig. 5.1.9), the color of the label is defined by the color of the pick.

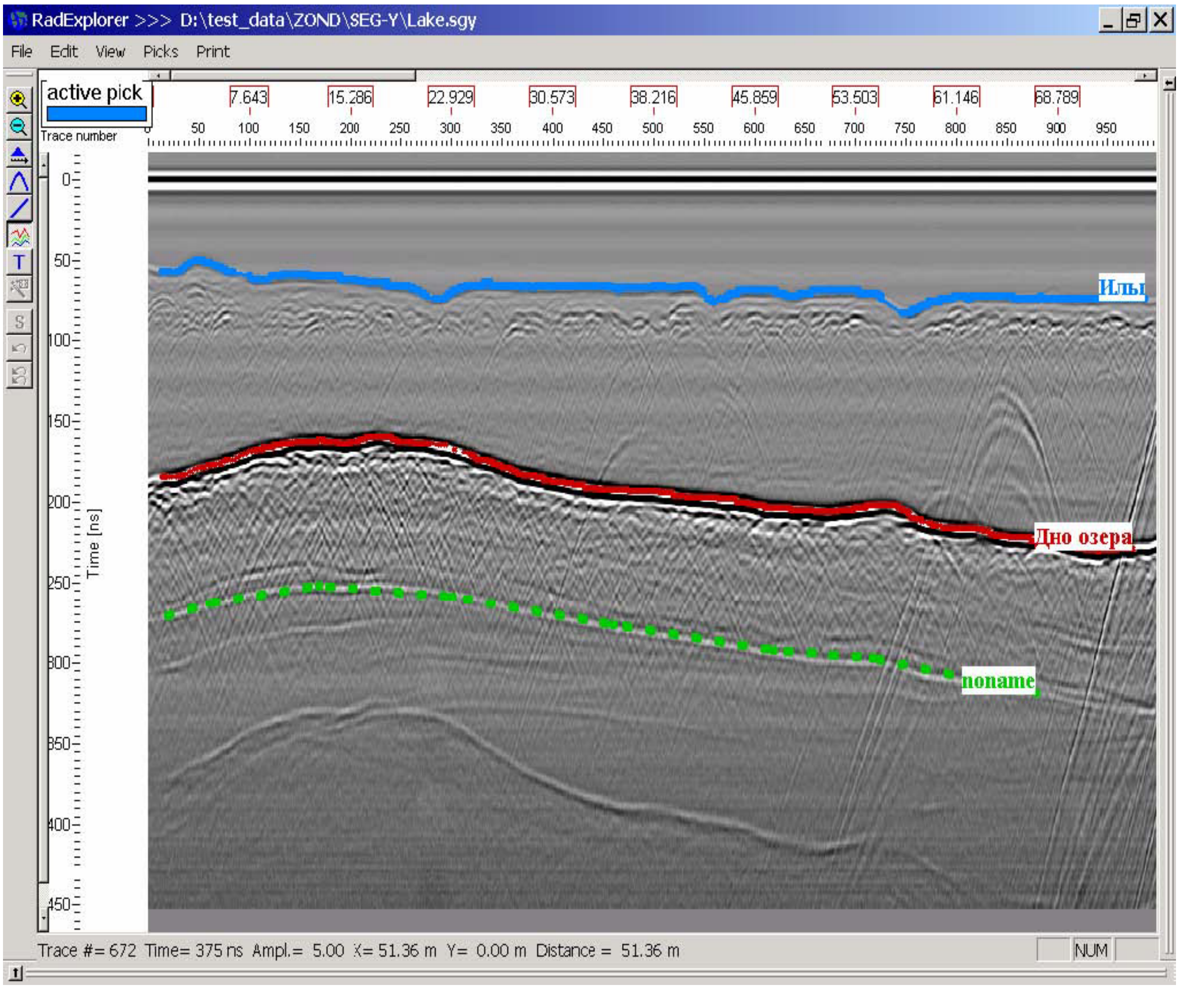



Fig. 5.1.9

To give a name to the active pick one should select the "**Name Active Pick…**" item of the menu. When doing this the dialog window (see Fig. 5.1.10) will open and you can enter the text there.

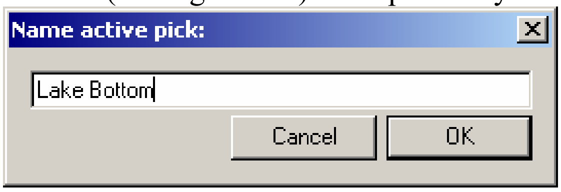



Fig. 5.1.10

The name will be saved while saving the whole pick with the help of the "**Save as…**" command. When you open the already existing pick from the file its name will be displayed on the screen (only if the "**Label picks**" option is activated).

Changing (selecting) of the active pick is accomplished in different ways: either by successive pressing the "Tab" key till the required pick becomes active (the active pick is indicated by the color in the picking mode icon in the upper left corner of the visualization window, see Fig.. 5.1.4) or by successive selection of the "**Next pick**" and the "**Previous pick**" commands in the "**Picks**" menu. In addition to the "Tab" key, one can switch from one pick to another by double-clicking the mouse button on the picking mode icon.

To exit picking mode click the  button on the toolbar again. All the picks that were created will remain on the screen but the commands for their editing and most of the commands from the "**Picks**" menu will become inaccessible.

## "Print" menu

This submenu contains the following commands:

- "**Print …**";
- "**Save image …**".

In order to print out the data select the "**Print…**" command. In this case the dialog box shown on Fig. 5.1.11 will open where one can chose the printer, specify the parameters and specify the scale for the image to be printed out.

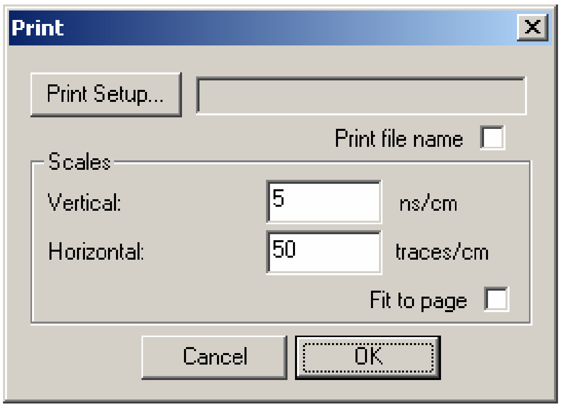



Fig. 5.1.11

When **OK** button is pressed *the whole dataset* will be printed. All elements of data analysis and interpretation being on the screen will be printed too: picks, model (with specified transparency), depth scale, etc.

**Print file name** option allows printing full name of the data file at the bottom left of every page.

When **Fit to page** option in **Scales** group in on, the print-out will be scaled to fit the currently selected page according to the settings of the printer driver.

*HINT*: In case you need to print out only a part of the radargram, you can delete the undesired traces with the help of *Trace edit* routine, print the data and than undo the routine.

The "**Save image …"** command allows the user to save the results of processing and interpretation on the disk as a bitmapped image in Windows Bitmap format (files with *.bmp extension). When this item of the menu is selected the dialog box for image parameters setting (see Fig. 5.1.12) will open.

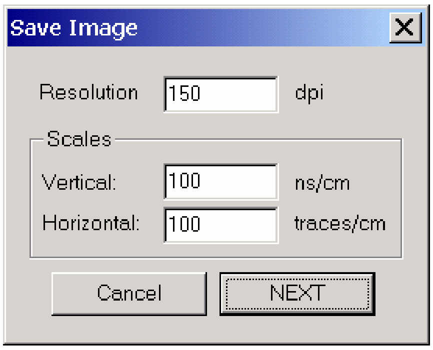



Fig. 5.1.12

Here one can specify the desired resolution (dots per inch) and the image scales. When pressing the “**Next…**” button the standard file dialog box will open. In this dialog box the user will be offered to choose the already existing file name or to enter a new name for a file where the image will be saved.

Similarly to the printing procedure, *the whole data file* will be exported into an image in the specified scale. All elements of data analysis and interpretation being on the screen will be exported too: picks, model (with specified transparency), depth scale, etc.

*HINT*: In case you need to export only a part of the radargram, you can delete the undesired traces with the help of *Trace edit* routine, print the data and than undo the routine.

## V.2. Toolbar

When launching the programme or opening a file, the toolbar (see Fig. 5.2.1) is located in the upper left part of the screen and gives the user an opportunity to zoom in and zoom out the image on the screen, to analyze the spectrum or wave propagation velocities, to switch on and off the picking mode, to place the text marks, etc.

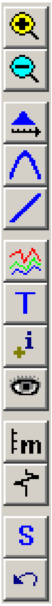


Fig. 5.2.1

Let us consider all the toolbar features.

## “Zoom in”

To zoom in the data fragment, select this button and then press the left mouse button in the corner of the fragment you would like to zoom in, and holding it pressed move the cursor to another corner of the fragment (see Fig. 5.2.2) and then release the button. The selected rectangular data fragment will be enlarged to the visualization window size (see Fig. 5.2.3).

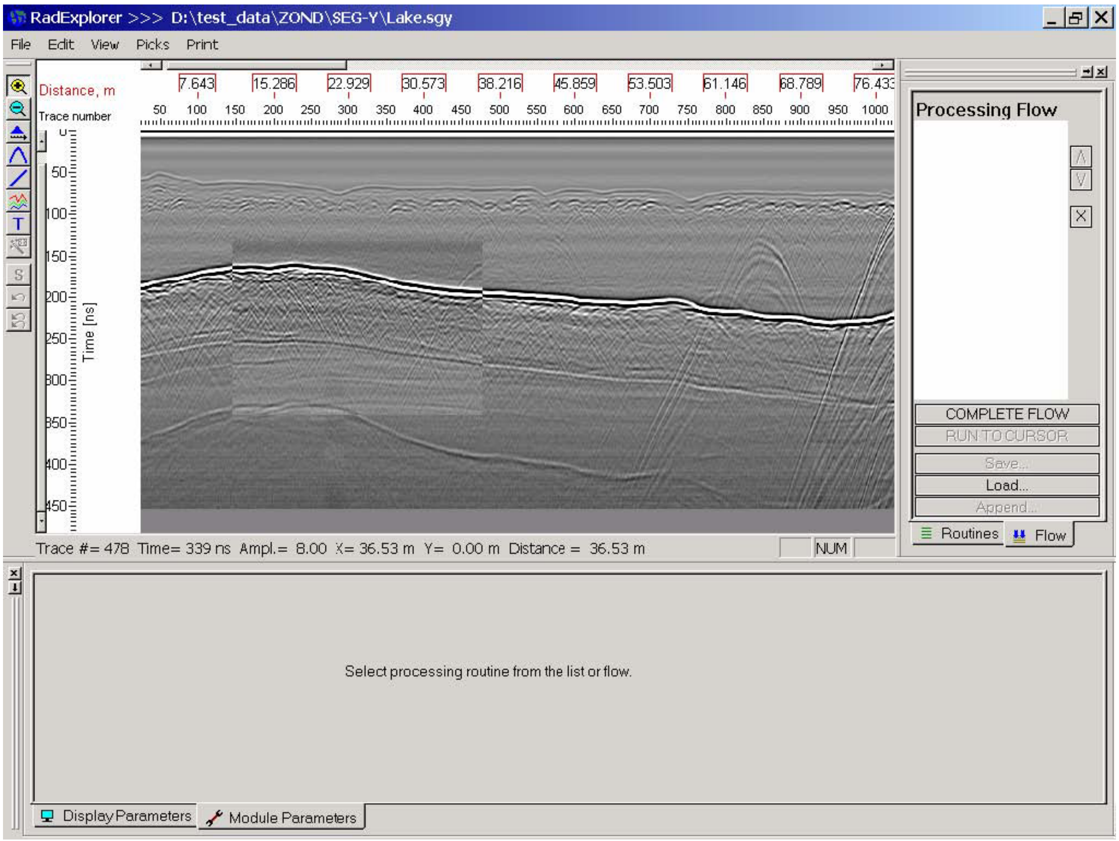



Fig. 5.2.2

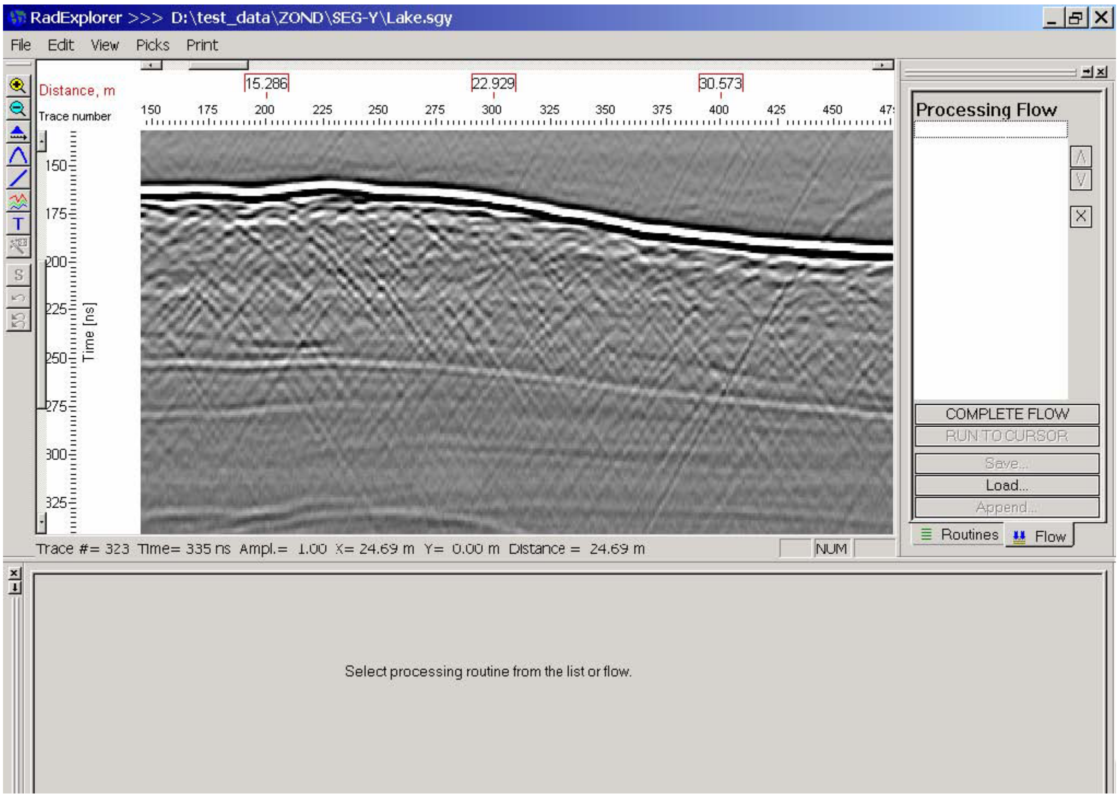



Fig. 5.2.3

The same result can be achieved by selecting the desired fragment on the horizontal or vertical scale axis at the upper or left edge of the visualization window respectively. To do this press the left mouse button at the beginning of desired fragment on the scale axis and, holding it pressed, move the cursor to the end of a fragment. The range selected on one of the scales will be enlarged to the screen size (the scale according to another axis will not change).

##  "Zoom out"

Pressing of the  button restores the viewing mode to the initially set scale (as on Fig. 5.2.2).

The user can zoom out an image to the initial scale by one of the axes by clicking the right mouse button on horizontal or vertical scale axis in the visualization window.

##  "Spectrum"

This command allows the user to view average frequency spectrum in a specified arbitrary window. The window is selected with the help of the mouse the same way as the zooming in is performed (see above). The average spectrum of the selected part of a radargram appears in a separate window (see Fig. 5.2.4).

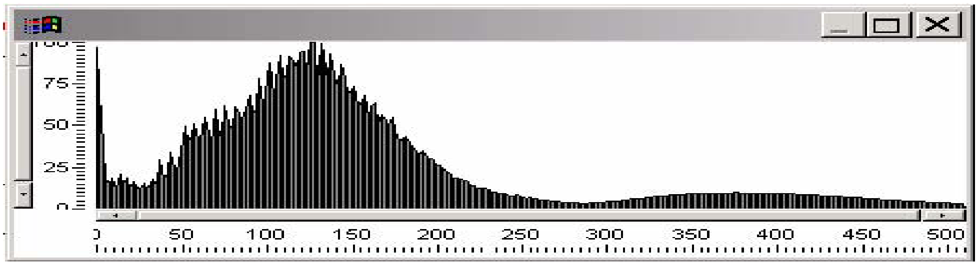


Fig. 5.2.4 Spectrum window. The horizontal axis dimensionality – MHz, the vertical axis dimensionality – percentage from the maximum.

To enlarge a spectrum fragment to the size of the whole window press the left mouse button at the beginning of the desired spectrum fragment on the horizontal axis and, holding it pressed, move the cursor to the end of the fragment. To return to the initial spectrum scale, click the right mouse button on the horizontal axis. The same way the spectrum fragment can be enlarged regarding vertical axis.

Several spectrum windows can be opened at the same time. To remove the spectrum window one should close the window.

##  “Hyperbola”

To determine the dielectric constant/wave propagation velocity within the medium the “*Hyperbola*” and the “*Line*” tools can be applied applied. After the  button was pressed a hyperbola-shaped curve of green or red color (see Fig. 5.2.5) will appear in the visualization window. This is theoretically calculated diffraction wave traveltime curve.

The hyperbola allows determination of diffracting object occurrence depth (i.e. the occurrence depth of the object that causes hyperbolic reflection on the radargram) and of average dielectric constant/ wave velocity within the medium.

At the bottom of visualization window the hyperbola parameters description field will appear.

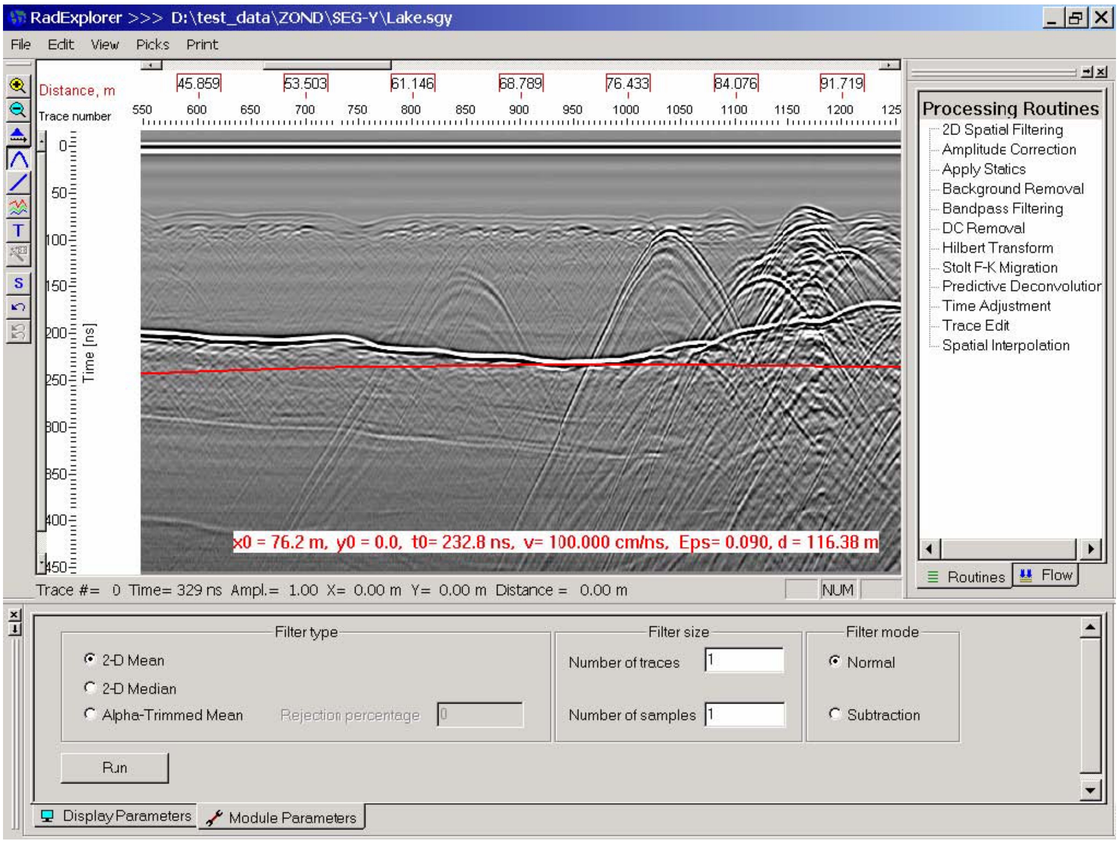



Fig. 5.2.5

One shall archive maximum possible matching of the theoretical hyperbola with a visible hyperbolic reflection on the radargram (see Fig. 5.2.6). By clicking the left mouse button one can set the hyperbola vertex at a cursor position point. By clicking the right mouse button one can define the steepness of hyperbola halves inclination. When the best possible match has been achieved, the hyperbola parameters displayed in the description string will give you the information about the target object:

- x0 – hyperbola vertex (i.e. target object) coordinate on X axis (if Y always equals 0 then it is the profile distance);
- y0– hyperbola vertex (i.e. target object) coordinate on Y axis (if the file geometry is defined. Otherwise y0=0 by default)
- Dist. – profile distance of the hyperbola vertex (i.e. target object)
- t0 – hyperbola vertex coordinate on vertical time scale;
- V – average electromagnetic wave propagation velocity within the medium;
- Eps – average dielectric constant of the medium;
- d – the depth hyperbola axis position (at velocity V), i.e. the target object depth.

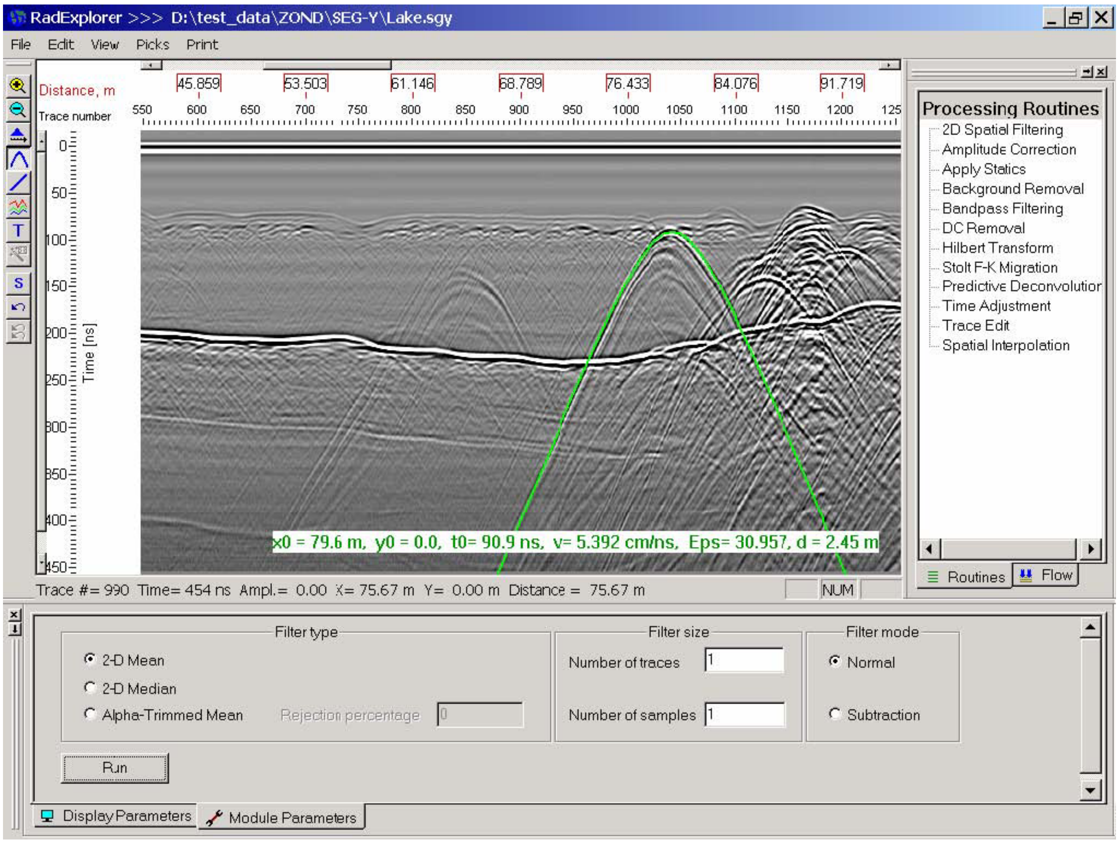



Fig. 5.2.6

It is known that electromagnetic wave propagation velocity within the medium V depends on dielectric constant of the medium ε:

$$V = C / \sqrt{\varepsilon} \text{ (where C – is velocity of light in vacuum).}$$

The dielectric constant ranges:

$$\varepsilon_{\text{in the air}} = 1 \leq \varepsilon \leq 80 = \varepsilon_{\text{in the water}}$$

and:

$$V_{\text{in the water}} = 3.3 \leq V \leq 30 = V_{\text{in the air}} \text{ respectively.}$$

If the obtained values go beyond the ranges, then the hyperbola becomes red on the screen.

*ATTENSION*: The accuracy of the object depth and dielectric constant/velocity determination is completely defined by accuracy of traces binding, i.e. by the accuracy of profile geometry definition! If you match the hyperbola to a real hyperbola reflection on the radargram accurately and the hyperbola color remains red it is most likely that the profile geometry was set incorrectly.

Pressing of the  button once again will close the “hyperbola” mode.

## “Line”

When the button is pressed, on the screen in the centre of visualization window the inclined line of green or red color will appear (see Fig. 5.2.7) – this is a fragment of theoretically calculated diffraction traveltime curve (“hyperbola” fragment). At the bottom of visualization window a line parameters description string will appear.

It is reasonable to apply this tool when only fragments or incomplete hyperbolas can be viewed on the radargram and the hyperbola vertex is impossible to determine.

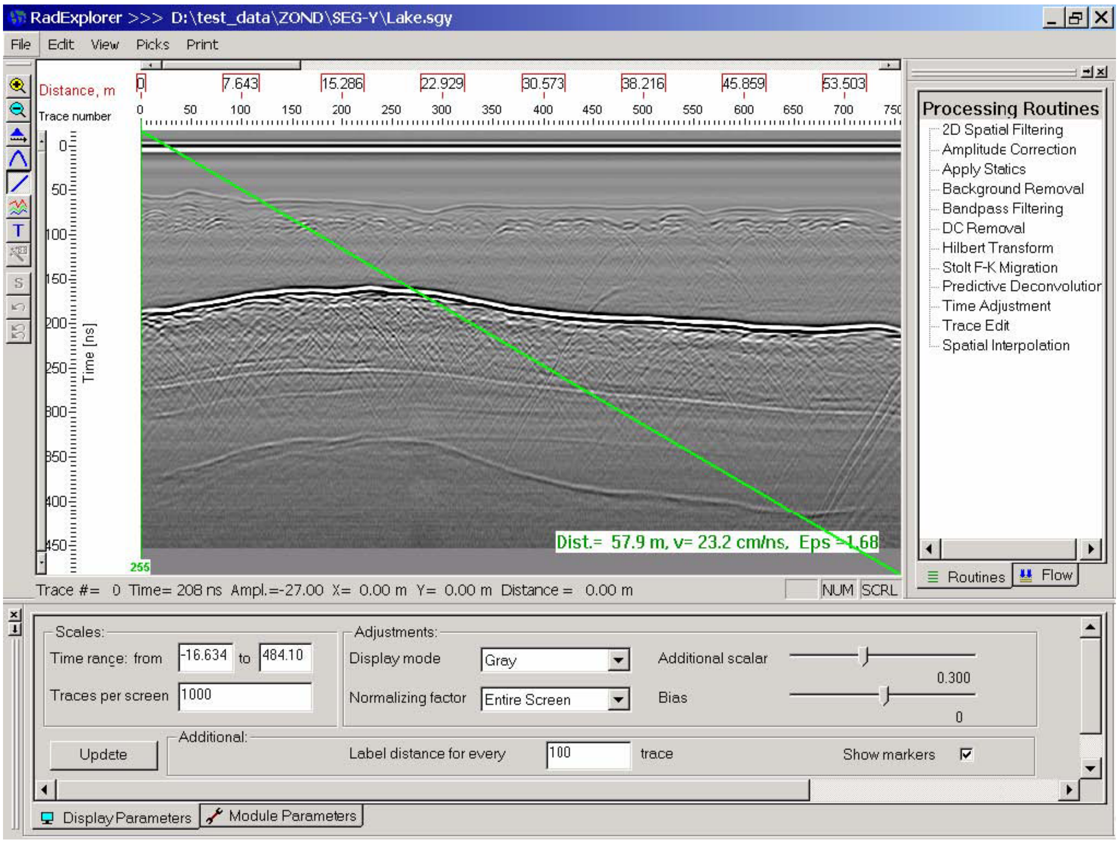



Fig. 5.2.7

By matching a theoretical line with obtained one we can achieve their maximum possible matching (see Fig. 5.2.8 for example). By clicking the left mouse button one defines the beginning of a line at cursor position point. By clicking the right mouse button one defines the steepness of line inclination. In this case the values of the following line parameters change:

- Dist – the across distance between the traces of the line beginning and end;
- V - average electromagnetic wave propagation velocity within the medium;
- Eps – average dielectric constant of the medium.

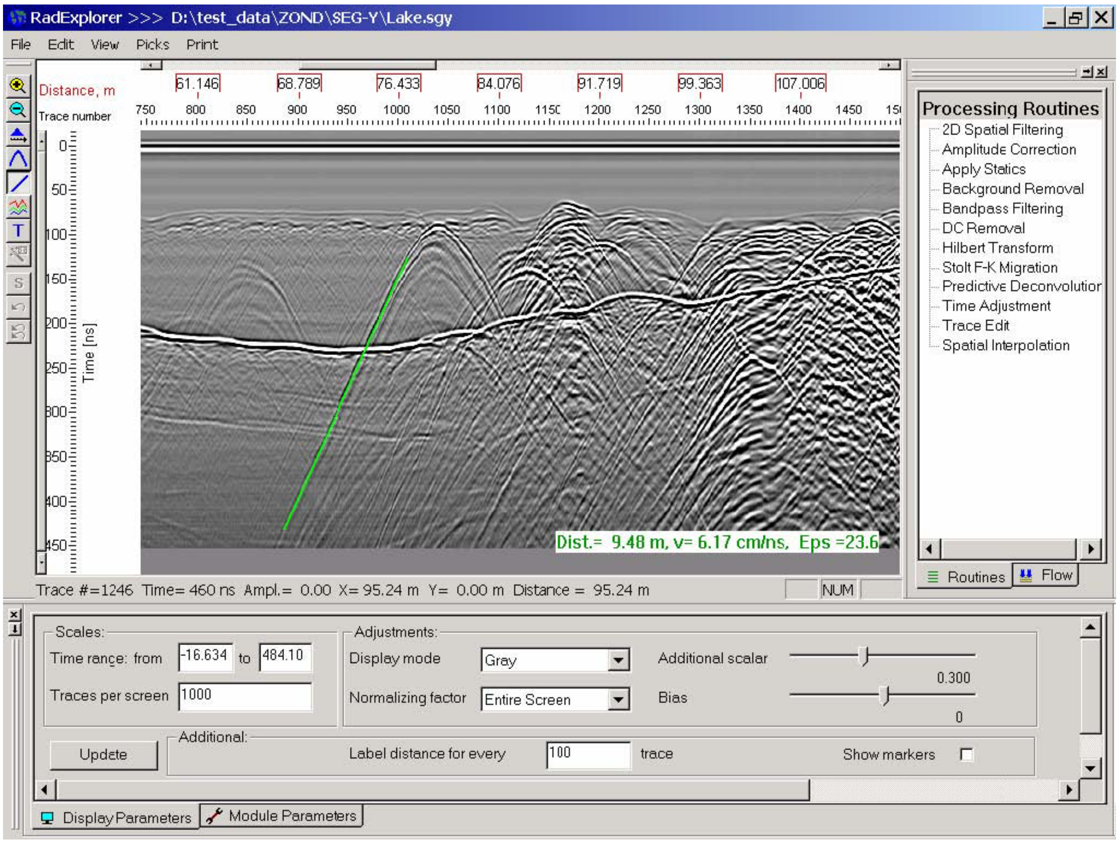



Fig. 5.2.8

If the obtained velocity values go beyond the $3.3 \leq V \leq 30$ range then the line on the screen becomes red.

*ATTENSION*: The accuracy of the object depth and dielectric constant/velocity determination is completely defined by accuracy of traces binding, i.e. by the accuracy of profile geometry definition! If you match the hyperbola to a real hyperbola reflection on the radargram accurately and the hyperbola color remains red it is most likely that the profile geometry was set incorrectly.

Pressing of the button once again will close the "*Line*" mode.

## "Picks"

Pressing the button switches on/off the picking mode. For detailed description of this mode see **"Picks/Picking mode" Menu** section.

## "Text mark"

This command allows the user to create a text mark – a label which is located in the visualization window. When pressing the button the window shown on Fig. 5.2.9 appears.

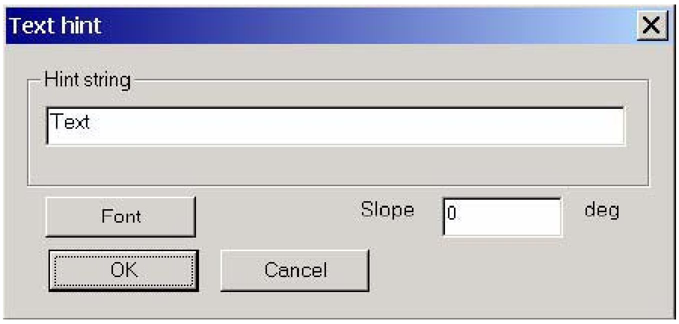



Fig. 5.2.9

In a “**Hint string**” field one should enter the text of the label; the “**Font**” button allows selection of font, its face and size. In the “**Slope**” field one should specify the row slope expressed in degrees (counterclockwise).

By repeated pressing of this button on the toolbar one can create several text marks and put them anywhere on profile.

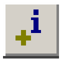
**The main commands for text marks editing:**

*To relocate* the created label on the screen one should capture it by the right mouse button and move it to a new place.

*To change the parameters* of the label (content, font, and slope) one should click the left mouse button on it.

*To delete the lable* one should click the left mouse button on it; erase the text in “**Hint string**” field and press “**OK**”.

When saving the file the signature will be saved too.

##  “Info mark”

This command allows to put a so-called informational mark (object) on any point of the profile. Press this button and then click on any point of the radargram. The dialog box shown on Fig. 5.2.10 will appear:

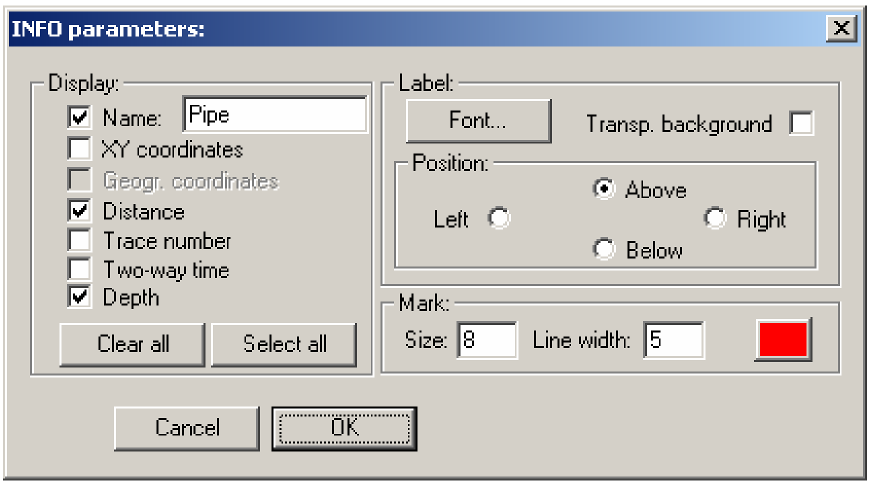



Fig. 5.2.10

In **Display** group on the left you can select the type of information about the point that you wish to display. You may select any combination of the specified types of information. Please, note that the depth (if selected) will be displayed in accordance with the current velocity model (see the "**Model tab**" section of this manual). The geographic coordinates are available for selection only when they are available in your data file.

The **Label** group on the top right allows you to specify the parameters of the text annotation: font, transparent or white background, position relative to the marked point (above, below, to the left, or to the right).

The **Mark** group on the bottom right allows adjusting of the appearance of the cross-mark, which will appear of the specified point of the radargram: its size, line width and colour.

When you select all the parameters press **OK**. A cross-mark with text annotation will appear, similarly to what is shown on Fig. 5.2.11.

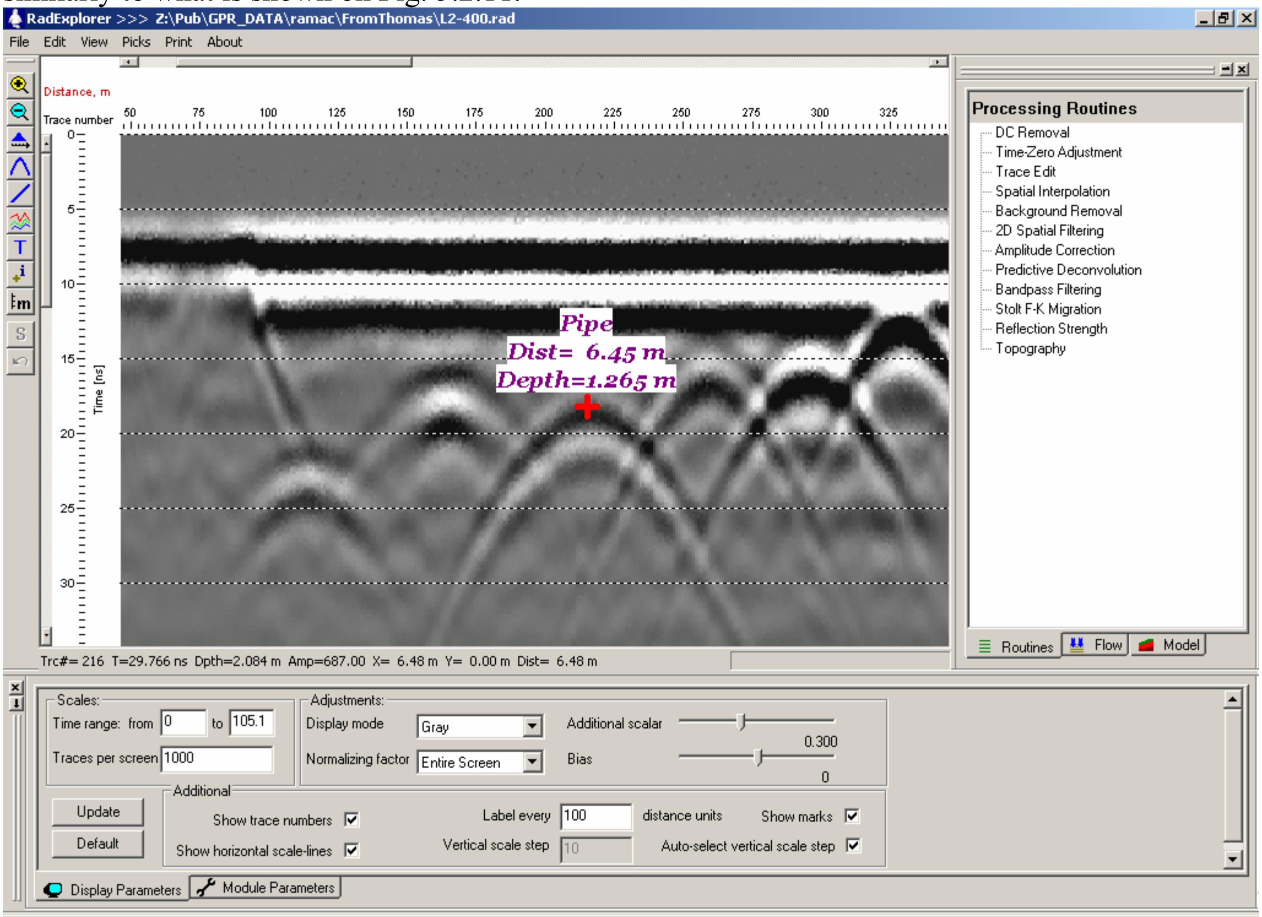



Fig. 5.2.11.

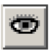
You can create as many info-objects as you want.

***Commands for info-object editing:***

After you have put an info object, its *parameters can be edited* by double-clicking on the corresponding cross-mark.

You can *move an info-mark to another point* by dragging the cross-mark with the right mouse button. After you drop the mark at a new location, the information text in the annotation will be updated automatically.

To *delete an info-object*, double click on it with the right mouse button. Alternatively, double click with the left mouse button to see its parameter dialog (see Fig. 5.2.10) and uncheck all the information types in the **Dispay** group (or press **Clear all** button). After you press **OK** the info-mark will disappear.

##  "Visibility of interpretation objects"

This button allows temporal 'switching off' of all interpretation elements created with the toolbar (i.e. picks, texts marks, and info marks). When the button is pressed, all interpretation elements become invisible both on the screen and for printing. Press the button again to restore their visibility.

##  "Trace"

Use this button to display a single wiggle trace above the radargram. Press the button and then click at the radargram where you would like the wiggle-trace to appear, thin green line markes zero-amplitude level of the trace (see fig. 5.2.12).

The vertical scale of the trace will be similar to that of the radargram, the width of the trace can be adjusted through the dialog window that is displayed upon right mouse-button click on the radargram.

The trace can be positioned on the radargam either by left mouse-button click, or by left and right arrow keys of the keyboard.

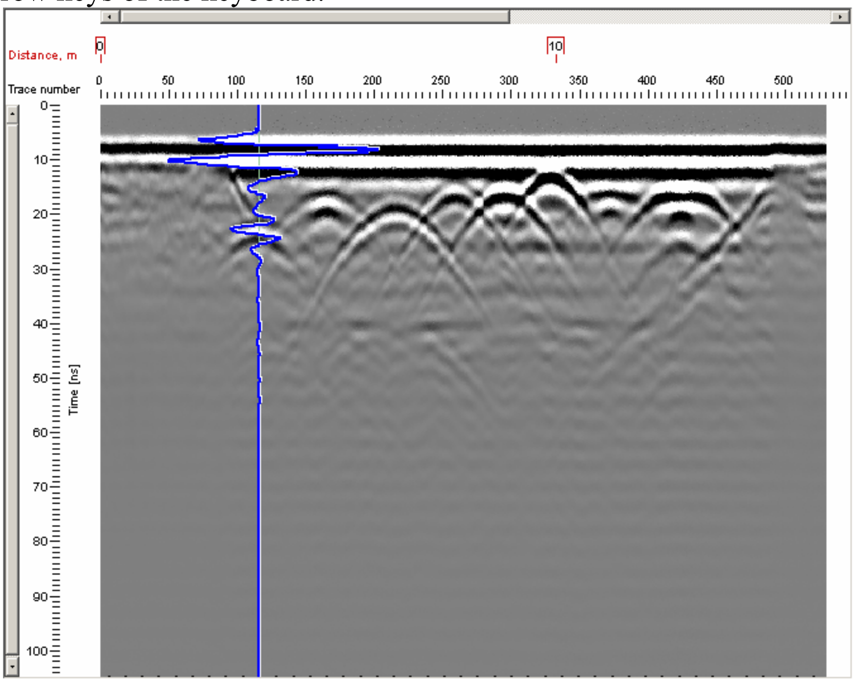



Fig. 5.2.12.

##  " Floating depth ruler"

If the radargram on the screen presented in a time scale then by pressing this button one can activate the "floating depth ruler" tool. This is a depth scale that shows the depth expressed in meters (or feet) in accordance with the current velocity model. Since the model can contain several layers with different wave propagation velocities, then at a constant time scale the depth scale might be nonlinear. In addition, since the outlines of the layers can change laterally, the different depth values can correspond to the same time values on different profile sections. The "floating depth ruler" takes this fact into account and shows the depth in the profile section selected by the user in accordance with the current velocity model (see Fig. 5.2.13).

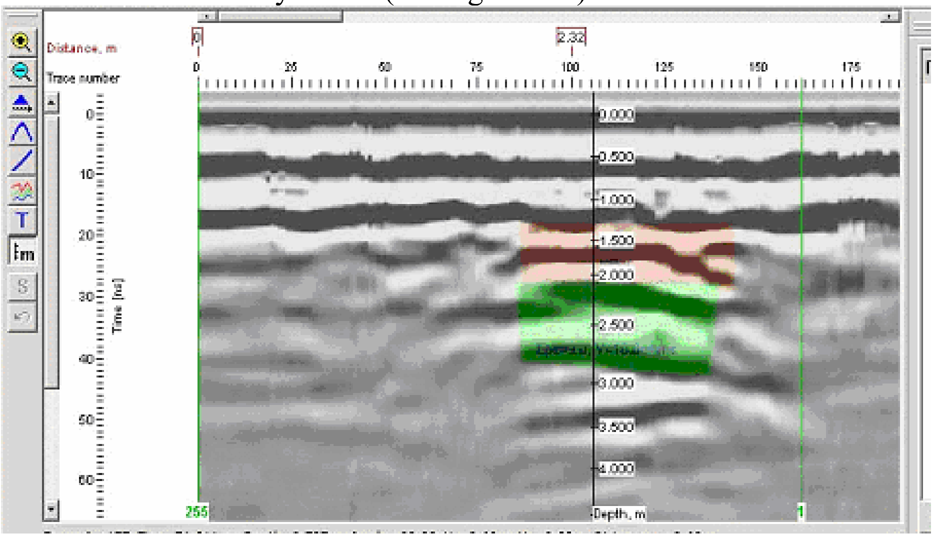



Fig. 5.2.13

To place the ruler at the required profile area one should click on the radargram by the left mouse button. The ruler will be placed on the trace corresponding to the current cursor position.

The ruler interval can be specified in a dialog box that will appear if one clicks on the radargram by the right mouse button (see Fig. 5.2.14).

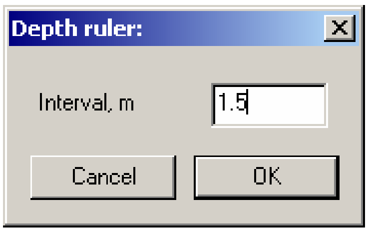
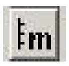



Fig. 5.2.14

To remove the ruler from the screen one should press the  button once again.

##  "Save changes in memory"

If after application of a processing routine one presses this button, the result of the routine operation will be saved in memory. After that, the repeated pressing of the "**Run**" button in the routine parameters window will result in routine application to the already *processed data* again. Otherwise, when one presses the "**Run**" button once again, the routine will be applied to the *unprocessed data* and that will make easier the routine parameters selection.

### "Undo the last applied routine"

If a processing routine was applied to the data, before saving the changes in memory one can undo the routine by pressing this button. After the changes have been saved (by means of pressing the S button or when switching to another routine) the button becomes inaccessible. After that the saved routine can be anytime undone through the "**Flow**" tab on the routine panel.

## *V.3. Visualization window*

The main data visualization window looks like on Fig. 5.3.1:

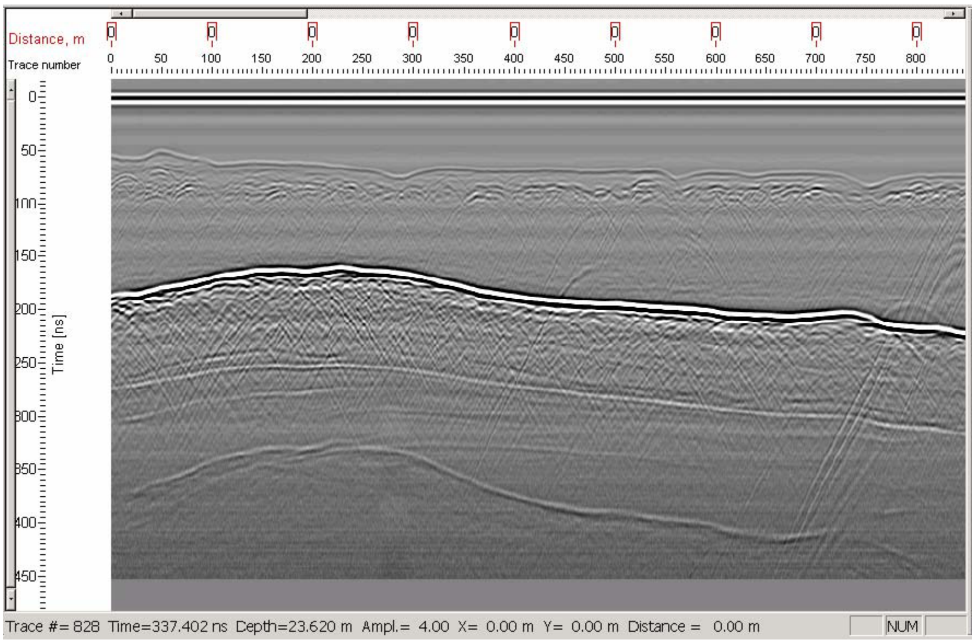



Fig. 5.3.1

The main window area is occupied by the GPR data radargram, displayed in a specified scale and image mode. On top and at the left there are horizontal and vertical scroll bars respectively.

Near the vertical scroll bar there is ***a vertical scale axis***. When the radargram is presented in time scale the electromagnetic wave two-way traveltime is expressed in nanoseconds ("Time [ns]"). After transformation into depth, the scale axis will show depth expressed in meters of feet ("Depth [m]" or ("Depth [ft]" ).

Under the horizontal scroll bar there is ***a distance scale axis*** ("Distance, m"). The numbers of this scale are marked out by red rectangular. Under it (directly above the data profile) there is ***a***

***trace number scale axis*** (“Trace number”).

In the status bar, parameters of the current cursor position in the window are being continuously displayed:

- “Trc #” – trace number;
- “T” – two-way wave travel time (in ns);
- “Dpth” – the depth expressed in meters in accordance with the current velocity model;
- “Amp” – signal amplitude;
- “X” – X coordinate in meters (if the geometry is determined);
- “Y” – Y coordinate in meters (if the geometry is determined);
- “Dist” – the profile distance in meters (if the geometry is determined).

If geographical coordinates are defined, they are also displayed in the status bar in addition to the above mentioned information.

## *V.4. Visualization and processing parameters panel*

The panel of visualization and processing parameters (see Fig. 5.4.1) contains two tabs:

- **“Display parameters”**;
- **“Module parameters” / “Velocity model editor”**.

### “Display parameters” tab

The “**Display parameters**” tab is a dialog box for data display parameters selection (see Fig. 5.4.1). This tab is always available when the panel of parameters is displayed on the screen.

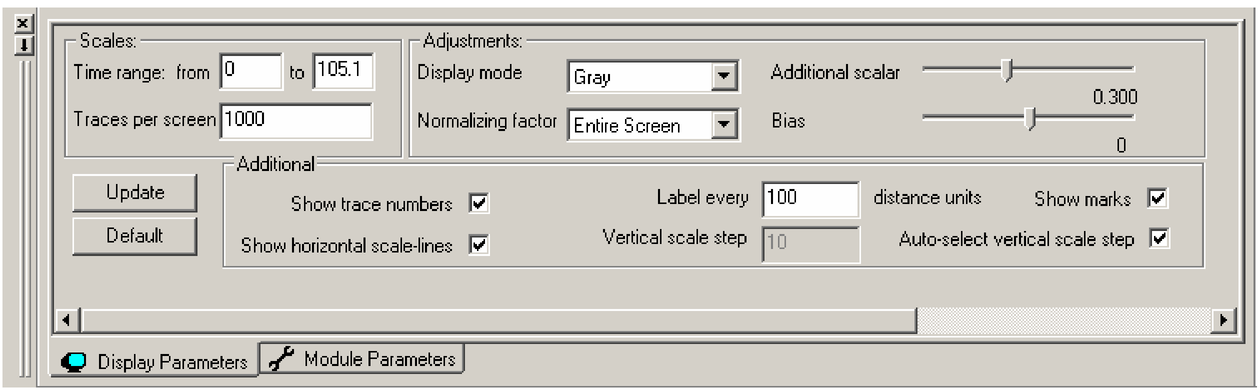



Fig. 5.4.1

In the “**Scale**” field one can specify the data displaying scales in the visualization window.

In a “**Vertically**:” field one should specify a vertical scale range: “**from**” – a starting value –

starting with this value the section will be displayed on the screen, “**to**” – a finite value – up to his value the section will be displayed on the screen. In a time mode of radargram displaying these values are to be specified in nanoseconds, after converting into depth – in meters or feet. The programme automatically restricts a finite value by the maximum value presented in the data. All the data can be viewed with the help of a scroll bar.

In the “**Traces per screen**” field the number of traces simultaneously displayed on the screen should be specified. If one specifies more traces than there are actually in the file the radargram will shrink and will not occupy the whole screen.

In the “**Adjustments:**” group the style and main parameters for visualization are specified.

In the “**Display mode**” field with a drop-down list one can chose the mode for trace displaying out of the following possible options:

- “WT/VA” – wiggle traces and variable amplitude (see Fig. 5.4.2);
- “WT” – wiggle traces (see Fig. 5.4.3);
- “VA” – variable amplitude (see Fig. 5.4.4);
- “Gray” – in gray scale (by default) (see Fig. 5.4.5);
- “R/B” – in red-white-blue scale (see Fig. 5.4.6).
- “Custom” – the traces are displayed in custom color scale.

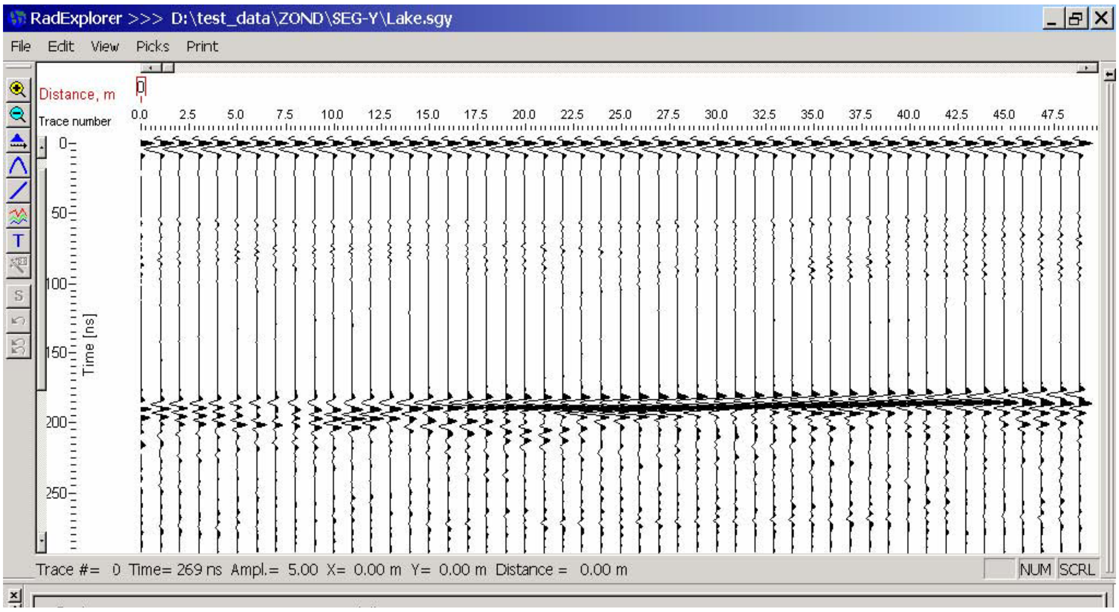



Fig. 5.4.2 – wiggle traces and variable amplitude

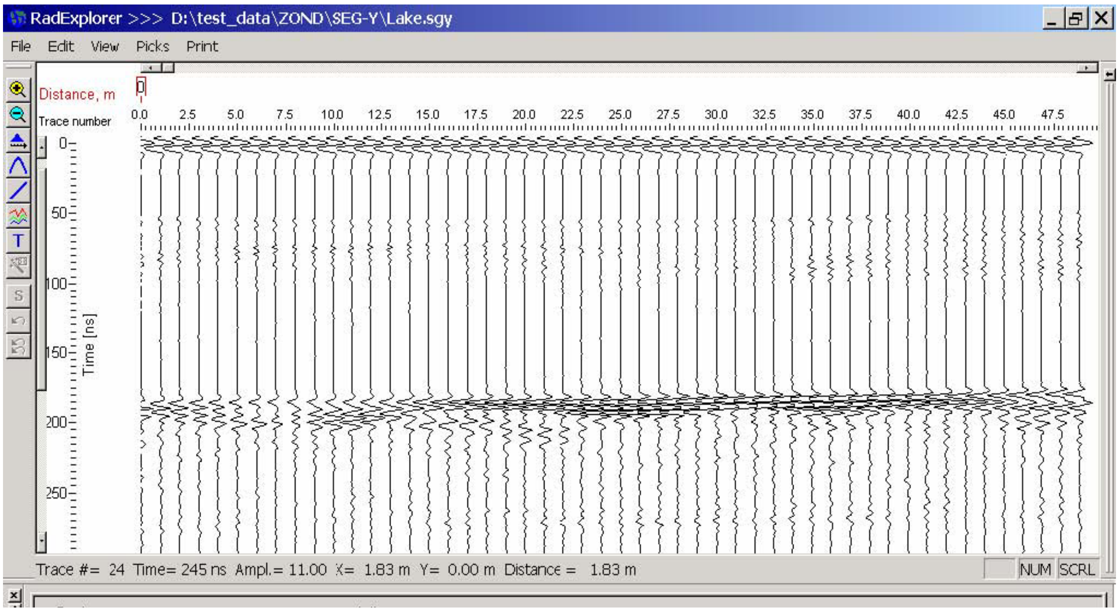



Fig. 5.4.3 – wiggle traces

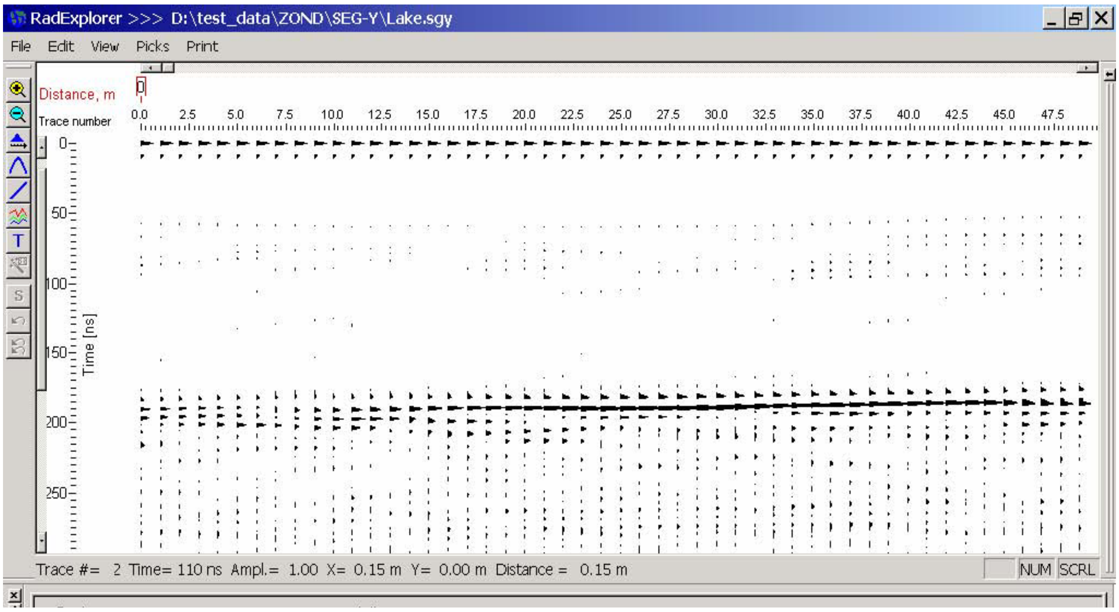



Fig. 5.4.4 – variable amplitude

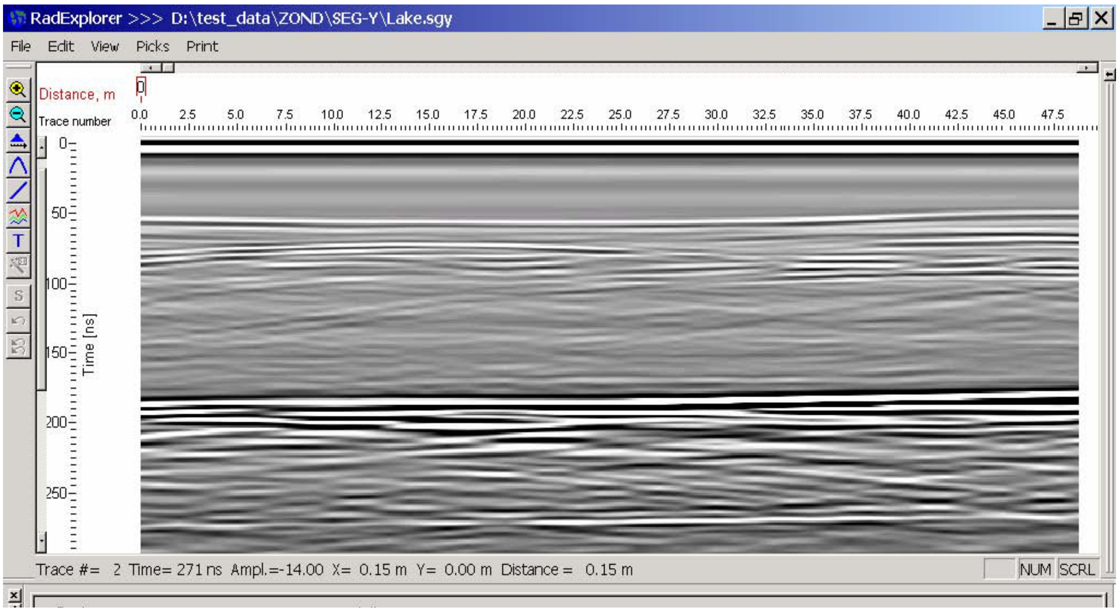



Fig. 5.4.5 – in gray scale

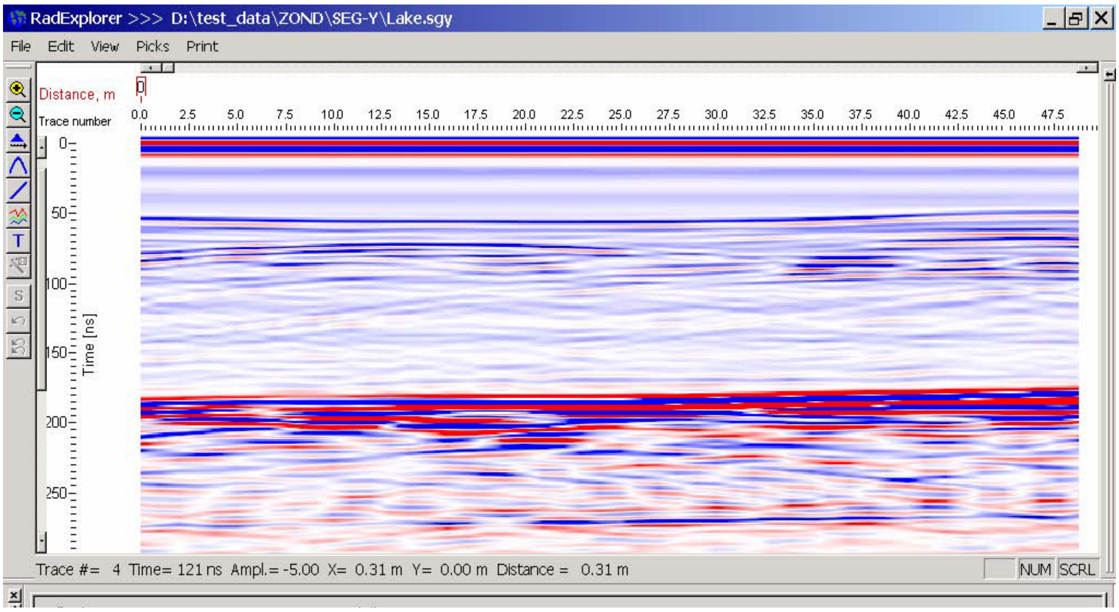



Fig. 5.4.6 – in red-white-blue scale

When “Custom” display mode is selected, first you will see the **Custom palette** dialog box (Fig. 5.4.7).

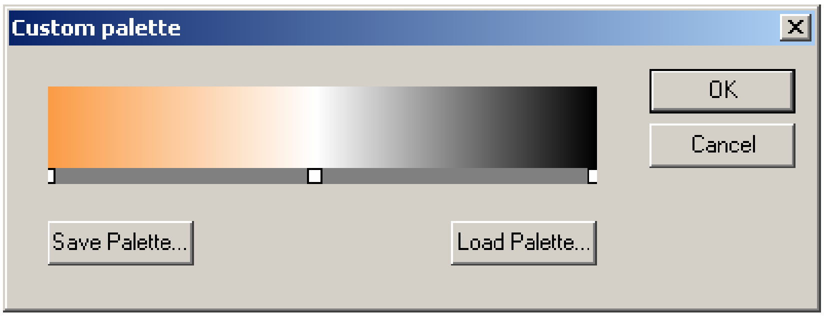


Fig. 5.4.7

Here you can load a palette from the hard disk (the default set of palettes is installed into the sub-folder PALETTES in the RadExplorer installation folder), customize the palette and save it onto hard disk.

***To customize the palette you can use the following commands:***

- *To move a palette point* simply drag it (the corresponding white square at the grey bar below the palette) with left mouse button and drop at a new position.

- *To add a new palette point* click on the desired position at the palette with left mouse button holding SHIFT key pressed. You will see a color selection dialog where you can chose the color of the new point.

- *To change the color of an existing point* double-click on it with the left mouse button. You will see a color selection dialog where you can chose the new color of the existing point.

- *To delete an existing point* click on it with the right mouse button.

In the "**Normalizing factor**" field with a drop-down list one can set the mode for normalizing the amplitude of traces displayed on the screen. Here are the options:

- "None" – traces are displayed as they are, without any additional normalizing (see Fig. 5.4.8).
- "Entire screen" – normalizes all the traces in the aggregate by means of trace amplitude division by the average absolute value of the amplitude of all traces (see Fig. 5.4.9);
- "Individual" – normalizes every trace individually by means of trace amplitude division by the average absolute value of the amplitude itself (see Fig. 5.4.10).

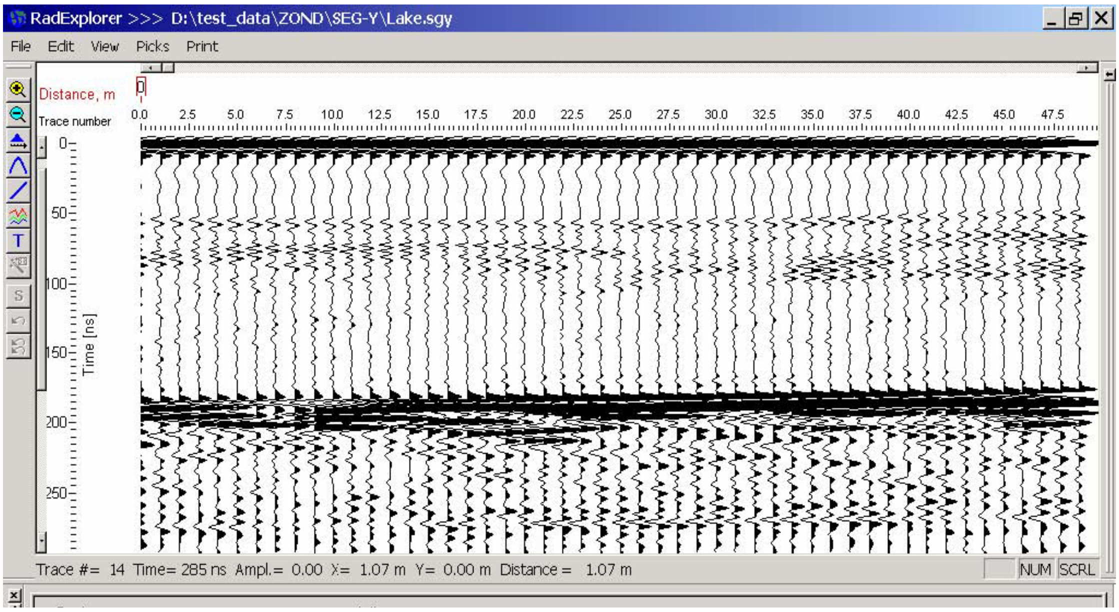



Fig. 5.4.8 Normalizing factor – "NONE"

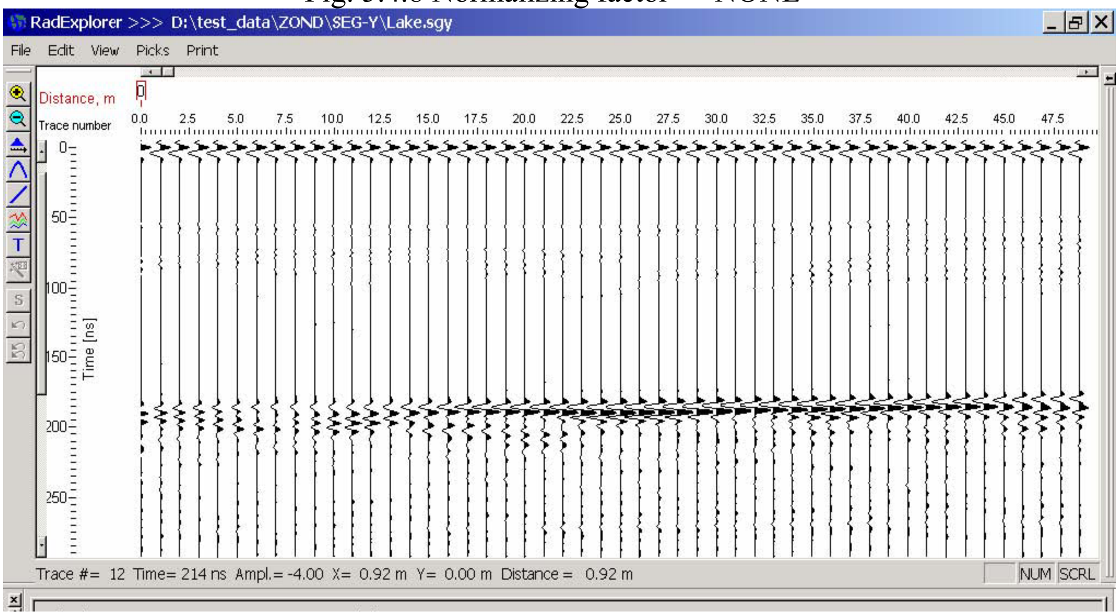



Fig. 5.4.9 Normalizing factor – "Entire screen"

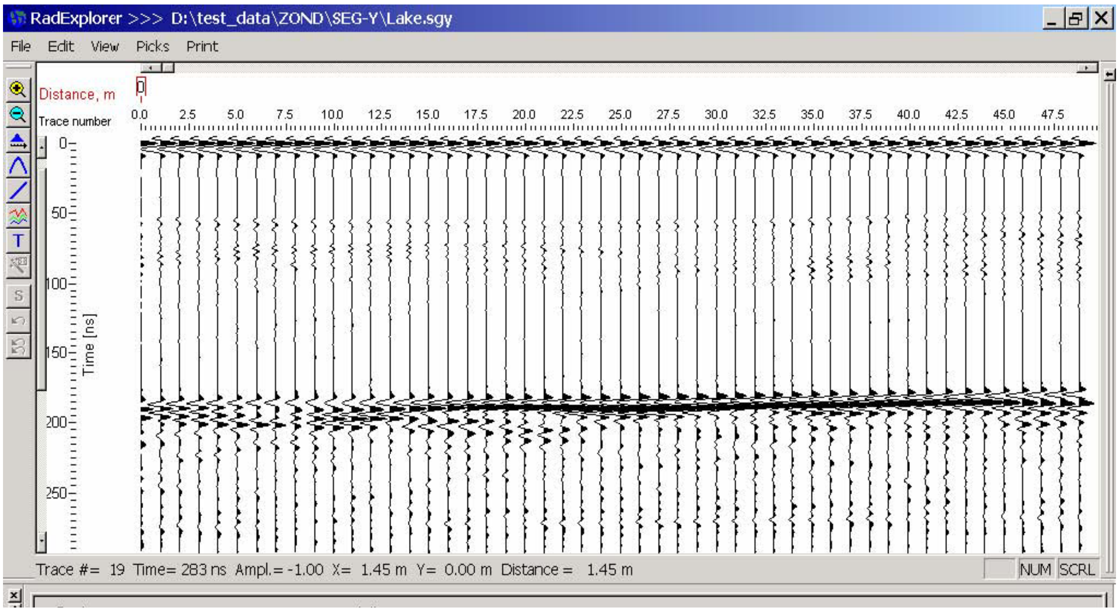


Fig. 5.4.10 Normalizing factor – "Individual"

The "**Additional scalar**" slider sets an extra factor value by which the trace samples before being displayed on the screen are multiplied. The current extra factor value is displayed to the right.

The "**Bias**" slider sets a value of average trace level bias from zero. In case the data is displayed in gray or red-white-blue scale the parameter changing results in zero signal level displacement in the color pallet. In case the traces are displayed in VA or WT/VA modes, the parameter variation results in black color level changing. The positive value will cause the displacement to the left from the zero line of the trace with enlarging the blacken area of the curve. The negative value corresponds to reducing of the blacken area of the curve. The current parameter value is displayed to the right.

In the "**Additional:**" field one can set up the additional adjustments.

**Show trace numbers** check-box allows to display or hide the *Trace number* horizontal axis.

**Show horizontal scale-lines** check-box control the presence of the dashed horizontal lines corresponding to major ticks of the vertical scale axis.

**Label every ___ distance units** – in this field you can indicate the tick interval of the *Distance* horizontal axis. The interval is either in meters or in feet depending on the programme settings.

**Show markers** check-box controls the displaying reference marks from the data file that have been set by the operator while recording. The reference marks are displayed on the screen as numbered vertical green lines (see Fig. 5.4.11).

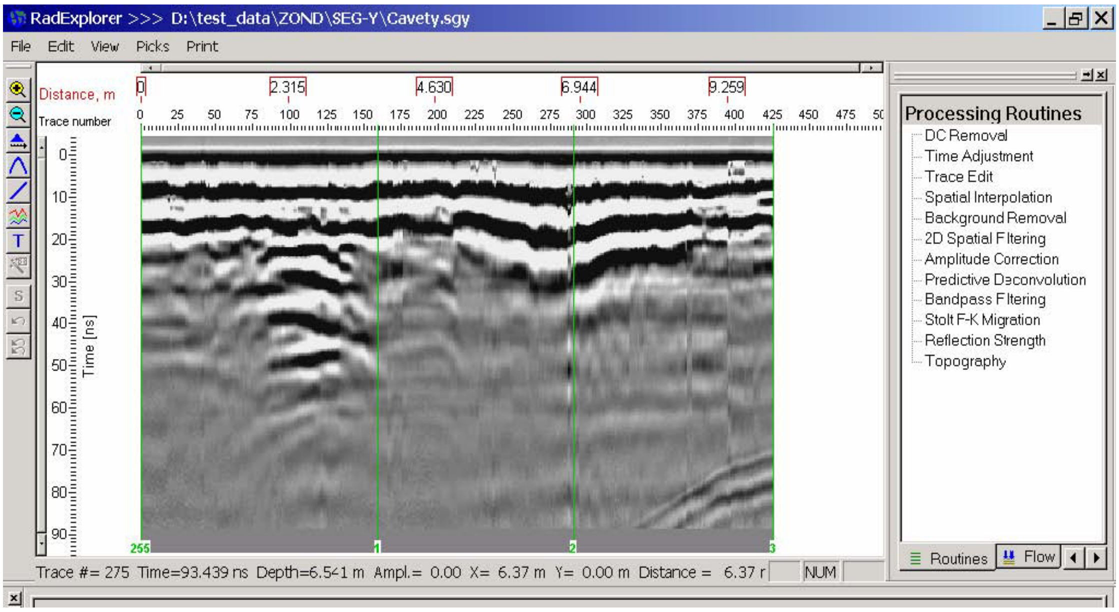



Fig. 5.4.10

**Auto-select vertical scale step** – if this check box is selected, the interval between the major ticks of the vertical axis is adjusted automatically. Otherwise, you can indicate the desired interval in **Vertical scale** step field.

To update the image on the screen in accordance with the selected parameters, click the "**Update**" button.

Upon exit, current display parameters are remembered and restored when you launch the application again. You can always click **Default** button to get back to the default display settings.

##  "Module parameters"/ "Velocity model editor" tab

In the "**Module parameters"/ "Velocity model editor**" tab you can specify the parameters for processing routines you have selected on the routines panel (on the "**Processing Routines**" tab or on "**Flow**" tab). In the same place the model editor interface is displayed in the model editing mode.

At first, right after the file has been opened the tab remains clear (see Fig.5.4.11). When selecting one or another processing routine the fields that allow changing the routine parameters appear in this window. We will discuss this tab in details in the section devoted to processing flow and velocity model panel.

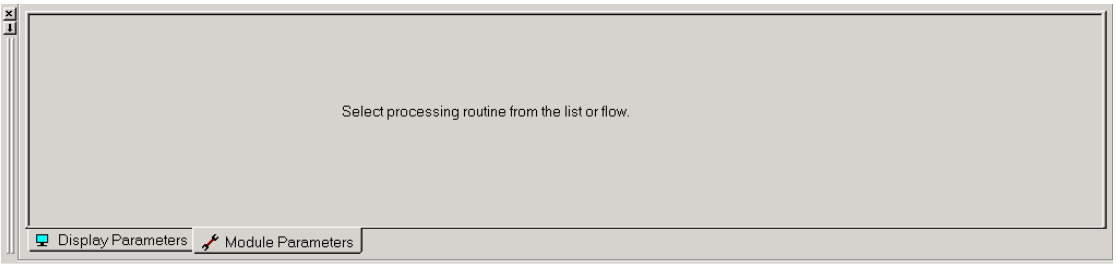



Fig.5.4.11

## *V.5. The panel of processing routines, flow and model*

The **RadExplorer** software contains standard routines of digital GPR data processing that are used for improving signal-to-noise ratio, increasing resolution, etc. With the help of a variety of techniques of signal processing one shall try to reduce the undesired waves and noise, remove or al least define them on the radargrams for not taking them into account during the interpretation.

The sequence of such transformations (routines) applied to the data file is usually called *data processing flow.*

The routines panel contains three tabs:

-  "**Processing Routines**"– contains the list of available signal processing routines;
-  "**Flow**"– allows the user to work with data processing history/flow;
-  "**Model**" – allows the user to create the profile model of the medium and to perform time to depth conversion of the radargram (i.e. recalculate time section into depth section).

Let us consider the features of every tab.

###  "Processing Routines" tab

The **RadExplorer** programme allows the user to apply the following routines to the data (see Fig. 5.5.1):

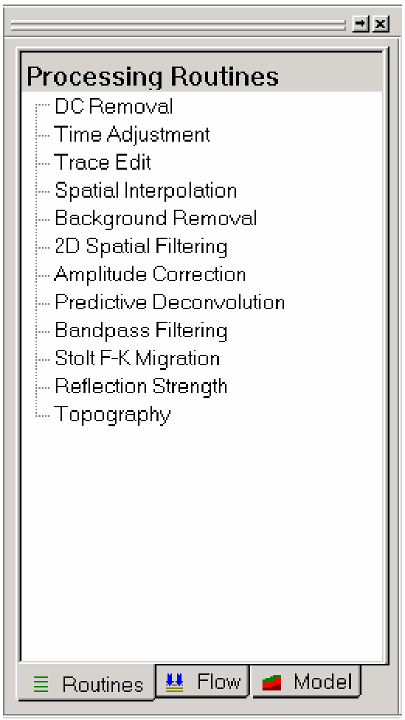



Fig. 5.5.1

The order of the routines in the "**Processing Routines**" tab corresponds to the order we suggest the user to apply them to the data. This does not mean that one must always apply all the routines or is not allowed changing the order of routines. One should apply only those routines that are required in every specific case. However, in case you are not sure concerning the order of operations you may try to apply them successively *top-down* ignoring those you definitely do not need. In case you do not know what parameters to choose then try the parameters set by default. In many of the cases you will manage to achieve the acceptable result. If the result is not acceptable you can always undo the whole processing or a part of it through the "**Flow**" tab and try it again but with different parameters.

Click the left mouse button on one of the routines from the list. The dialog box with its parameters will display in the "***Module parameters***" tab (Fig. 5.5.2).

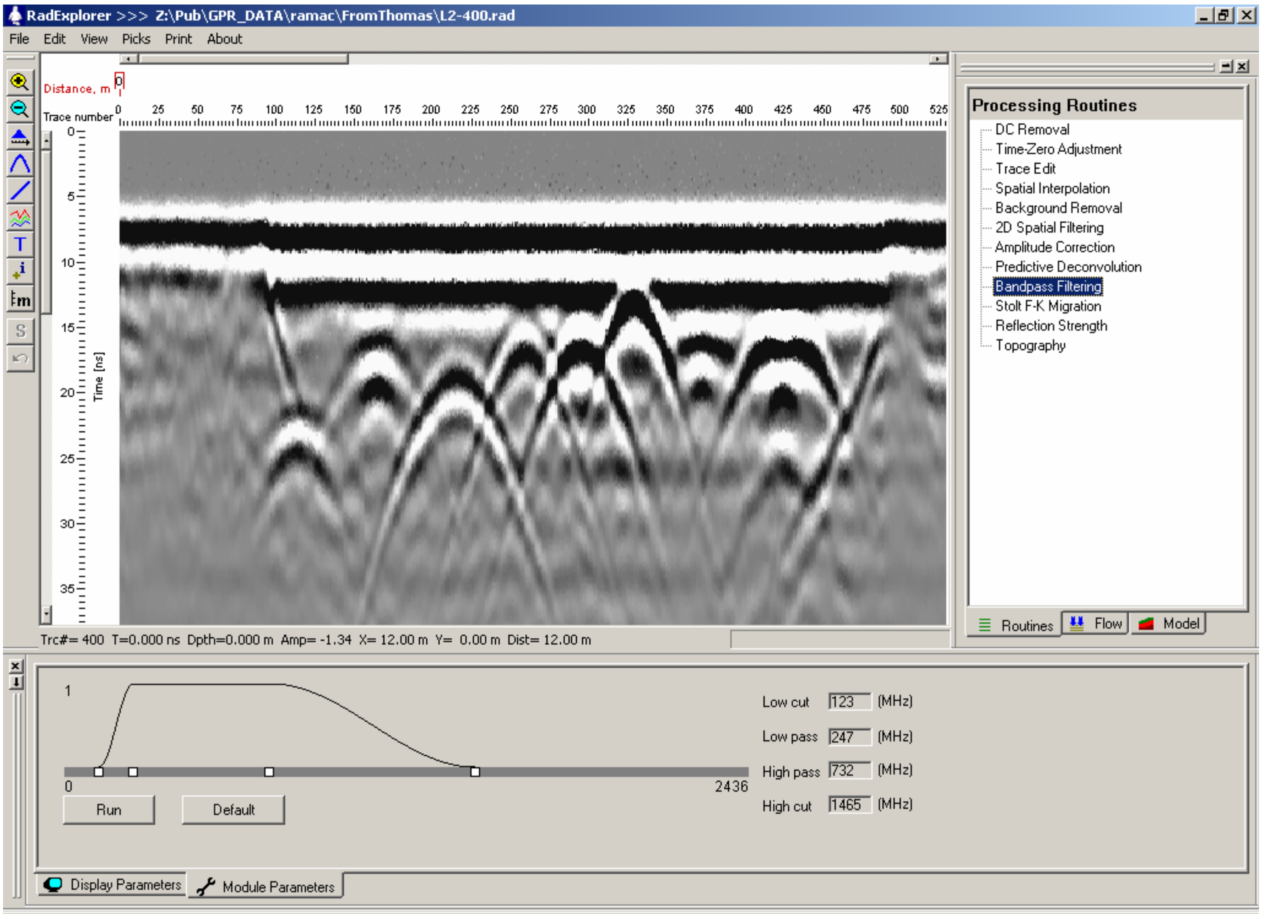
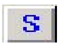
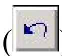



Fig. 5.5.2

After you have specified the parameters, click the "**Run**" button in the "***Module parameters***" tab. The routine will be applied to the data. If the result is acceptable for you, then proceed to the next routine. If not, then change the parameters and click the "**Run**" button once again – as long as you have not saved the changes in memory (with the help of  button on the toolbar, when moving to the next routine, when switching to the "**Flow**" or "**Model**" tab) the processing will be applied to the same data, i.e. to the data used before application of this routine. You can display the initial data on the screen by canceling the routine by pressing the undo button () on the toolbar (before the changes have been saved in memory). After the changes have been saved you can undo the whole processing or any part of it through the "**Flow**" tab.

Whenever you wish to get back to original settings of the routine, press **Default** button, which is available for most of the processing modules.

When the acceptable result of the routine application is achieved you can select the next routine from the list. In case you have not saved the result of the processing in memory the dialog box shown on Fig. 5.5.3 will appear.

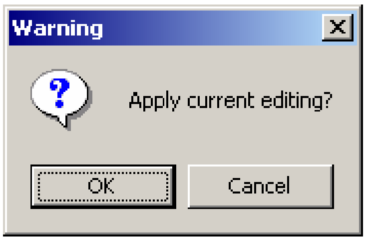


Fig. 5.5.3

If you press “**OK**” button, the result of the application of the previous routine will be saved in memory. Pressing the “**Cancel**” button will result in cancellation of the previous routine. Whatever is the case, the parameters of the newly selected processing routine will appear in the second tab of the parameters panel.

## Processing routines of the RadExplorer software

### *DC Removal*

The “*DC Removal*” routine is meant to remove constant component of the signal in case there is one. Single-click the left mouse button in a corresponding line of the routine list to see the dialog window for adjusting the routine parameters in the “**Module parameters**” tab (see Fig. 5.5.4):

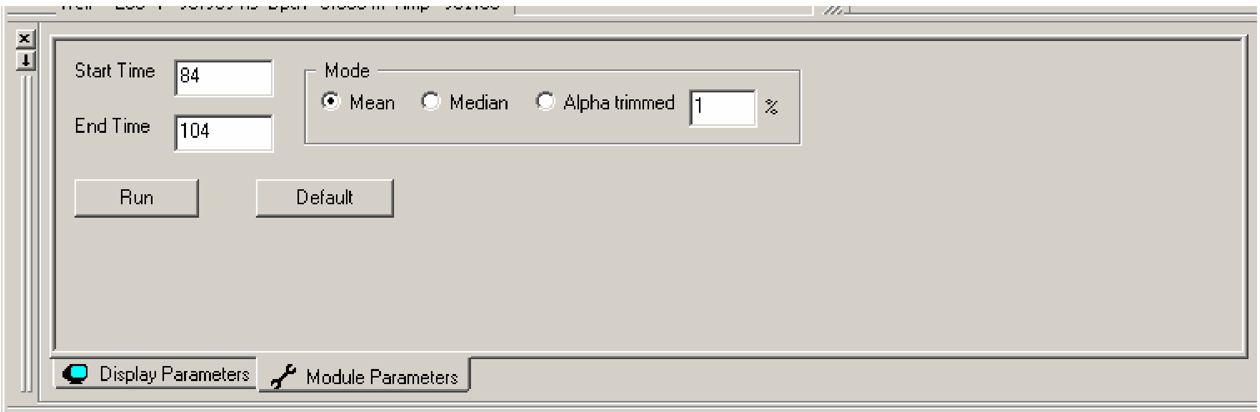



Fig. 5.5.4

In the “Start time” and “End time” fields one should specify the limits of the time range within which the constant component evaluation will be accomplished.

In the “Mode” fild, the mode of constant component evaluation should be specified:

- “Mean” – the arithmetic mean value of the samples within the specified time range is taken as a constant component value;
- “Median” – the median mean value of the samples within the specified time range is taken as a constant component value;
- “Alfa trimmed” - the mean value of the samples excluding the specified percentage of the lowest and the highest values within the specified time range is taken as a constant component value;

In the upper part of the Fig. 5.5.5 the initial radargram is shown; in the lower part – the radargram after the DC removal.

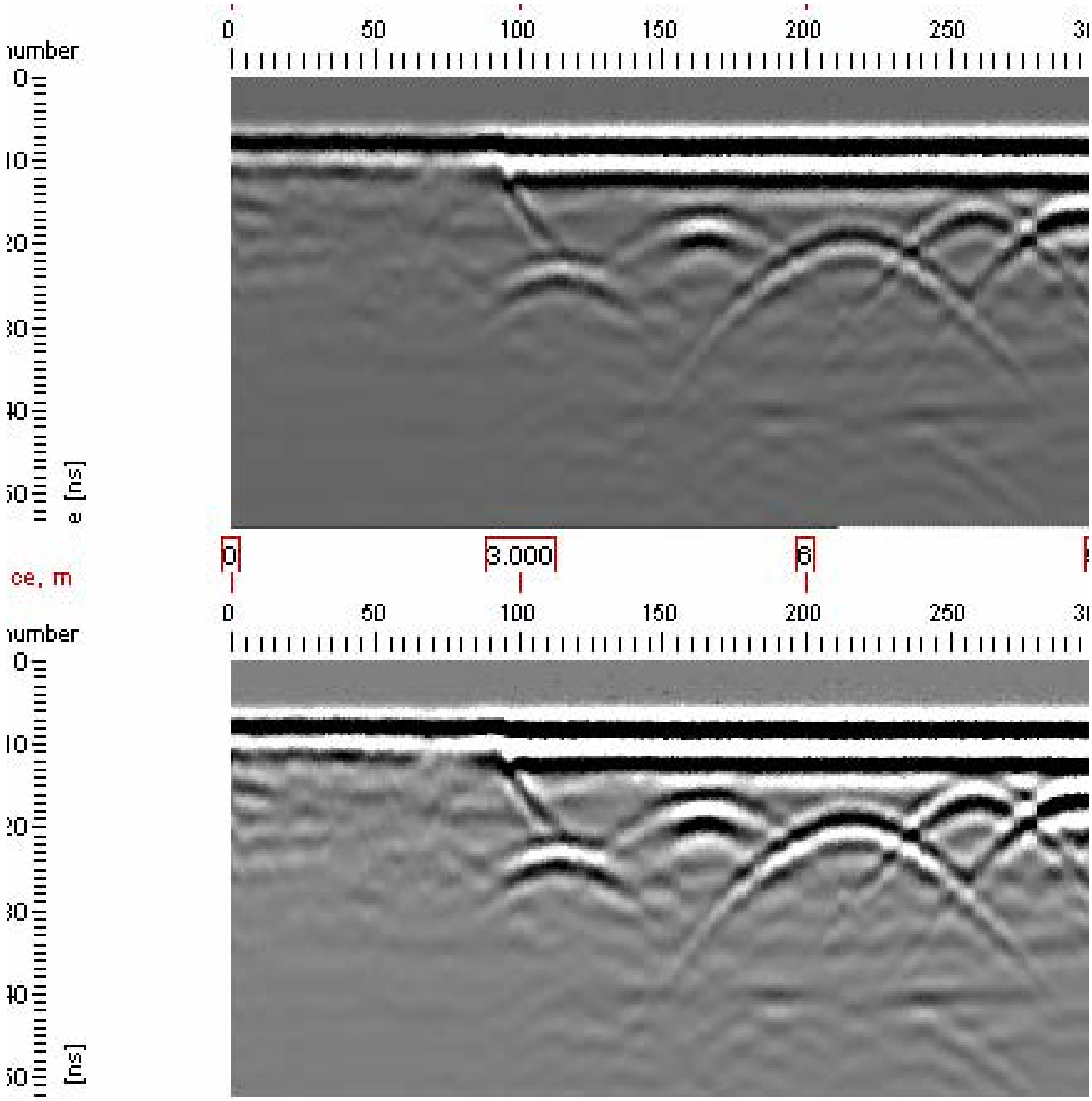



Fig. 5.5.5 – before and after the “DC Removal” routine application

## *Time Adjustment*

The “*Time Adjustment*” routine (see Fig. 5.5.6) is meant for adjusting the zero-point of the vertical time scale to the time-zero, i.e. the moment when the wave has actually left the emitting antenna.

When moving the “First-arrival time (ns)” slider one can set the direct wave arrival time. At the same time in the visualization window the red horizontal dotted line that defines the first-arrival time position is moving.

In the “Distance between the antennas” line one should specify the distance between receiving and transmitting antennas; in the “Velocity” line – the effective electromagnetic wave propagation velocity in the medium between receiving and transmitting antennas.

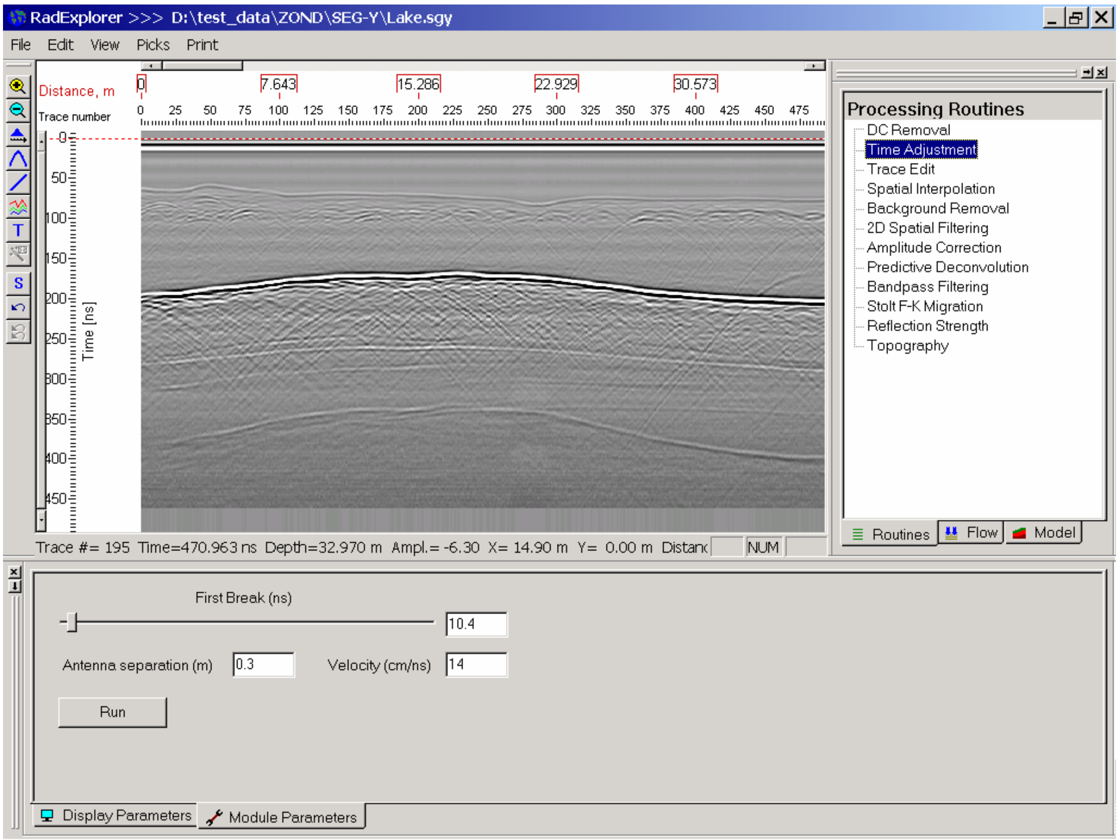



Рис. 5.5.6

## *Trace Edit*

The “*Trace Edit*” (see Fig. 5.5.7) module allows the user to exclude the invalid and undesired traces and record intervals from the initial record.

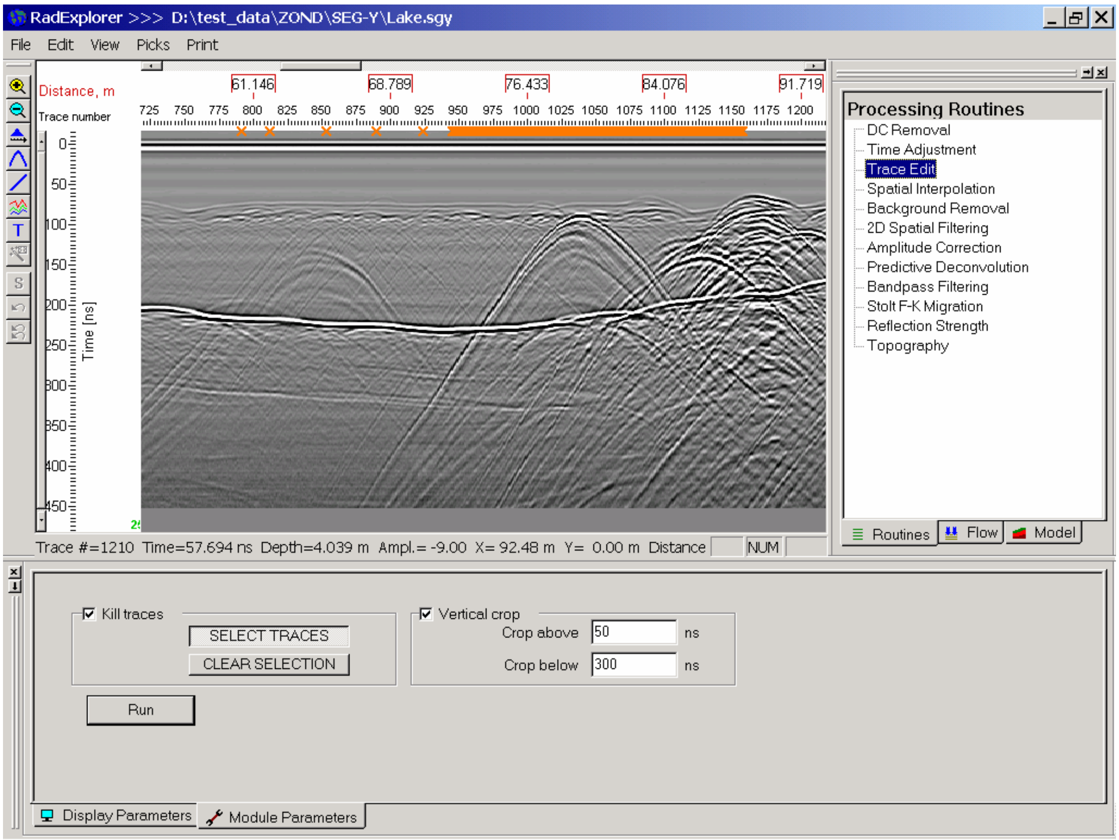



Fig. 5.5.7

In the parameters window one should specify the following parameters.

When the "Trace Edit" button has been pressed the user can select the undesired traces to be deleted in the "Kill traces" field.

You can select traces that should be deleted individually or by groups. To delete a single trace one should click the left mouse button on it in the visualization window (the active trace number is always displayed in the status bar). In the upper part of the selected trace the orange cross will appear. In order to select a group of traces one should press the left mouse button at the beginning of the desired section and holding it pressed move the mouse pointer to the end of the fragment to be deleted. To cancel the trace selection you should do the same but use the right mouse button. If one clicks the "Clear selection" button it will canel the selection of all traces.

In the "Vertical crop" group in the "Crop above" and the "Crop below" lines the user can specify the time value the trace sections over and below which will be deleted.

After the "**Run**" button has been pressed all the selected traces and record sections will be deleted.

*ATTENTION*: When deleting the traces, the binding of the rest of the traces DOES NOT change and remains *correct*!

### *Spatial interpolation*

The "*Spatial interpolation"* routine is meant for data recalculation on a regular profile interval by means of interpolation of the traces in horizontal direction. The distance between the traces can also be recalculated to a larger or smaller interval.

Usually, application of this routine is required while processing the data obtained with time trigger, i.e. without distometer. In this case when one specifies the desired profile interval (ideally it should not differ from the real average interval that much) it becomes possible to correctly recalculate the data on the even interval. Due to the same reason it also makes sense to apply this routine when part of traces from the middle of the profile has been killed with the help of "*Trace edit*".

To accomplish the routine, enter new desired distance between traces (profile interval) expressed in meters (or feet) in the dialog box (see Fig 5.5.8) of the routine in the "New dL:" field.

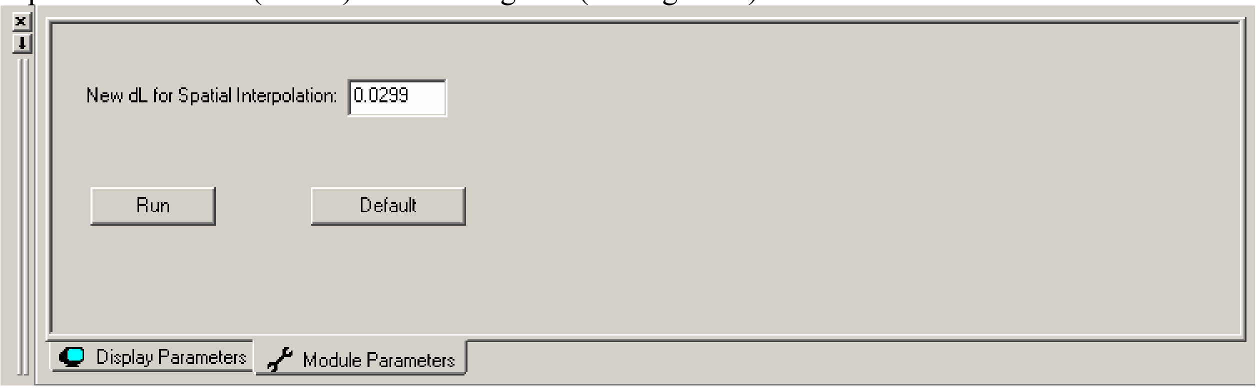



Fig 5.5.8

*ATTENTION:* When applying this routine the trace binding is being *correctly* recalculated!

### *Background Removal*

The most intensive signal received by the ground-penetrating radar is the signal that arrives directly from the transmitting antenna – a direct wave. The application of *Background removal"* routine is necessary when the instrument noise blocks up the desired signal. The essence of this technique is in subtraction of the mean trace determined in the window with fixed size running along the profile from the whole set of traces. It results in removing a constant instrument component from recording. But in this case one should always keep in mind that along with the undesired signal, the signal from real sub-horizontal boundaries can be lost. The only parameter of this routine defines the running window size (Strong = the narrow window, Weak = the wide window) (see Fig. 5.5.9).

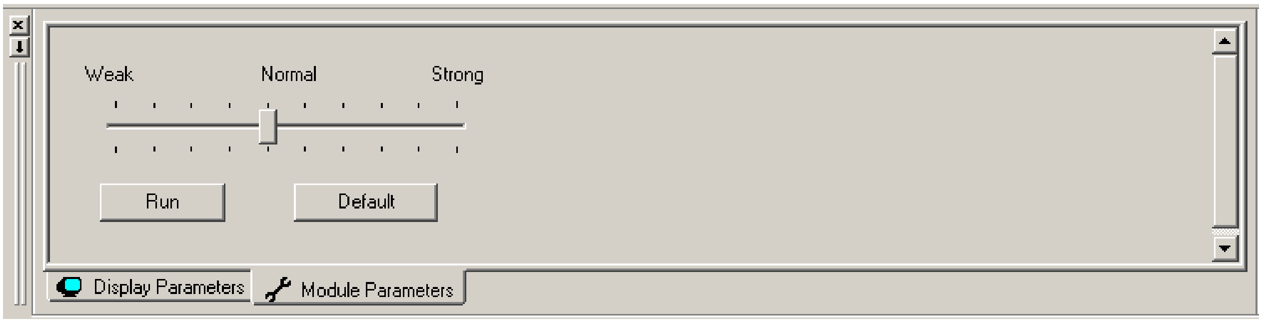



Fig. 5.5.9

The radargram before and after the background removal is shown on Fig. 5.5.10.

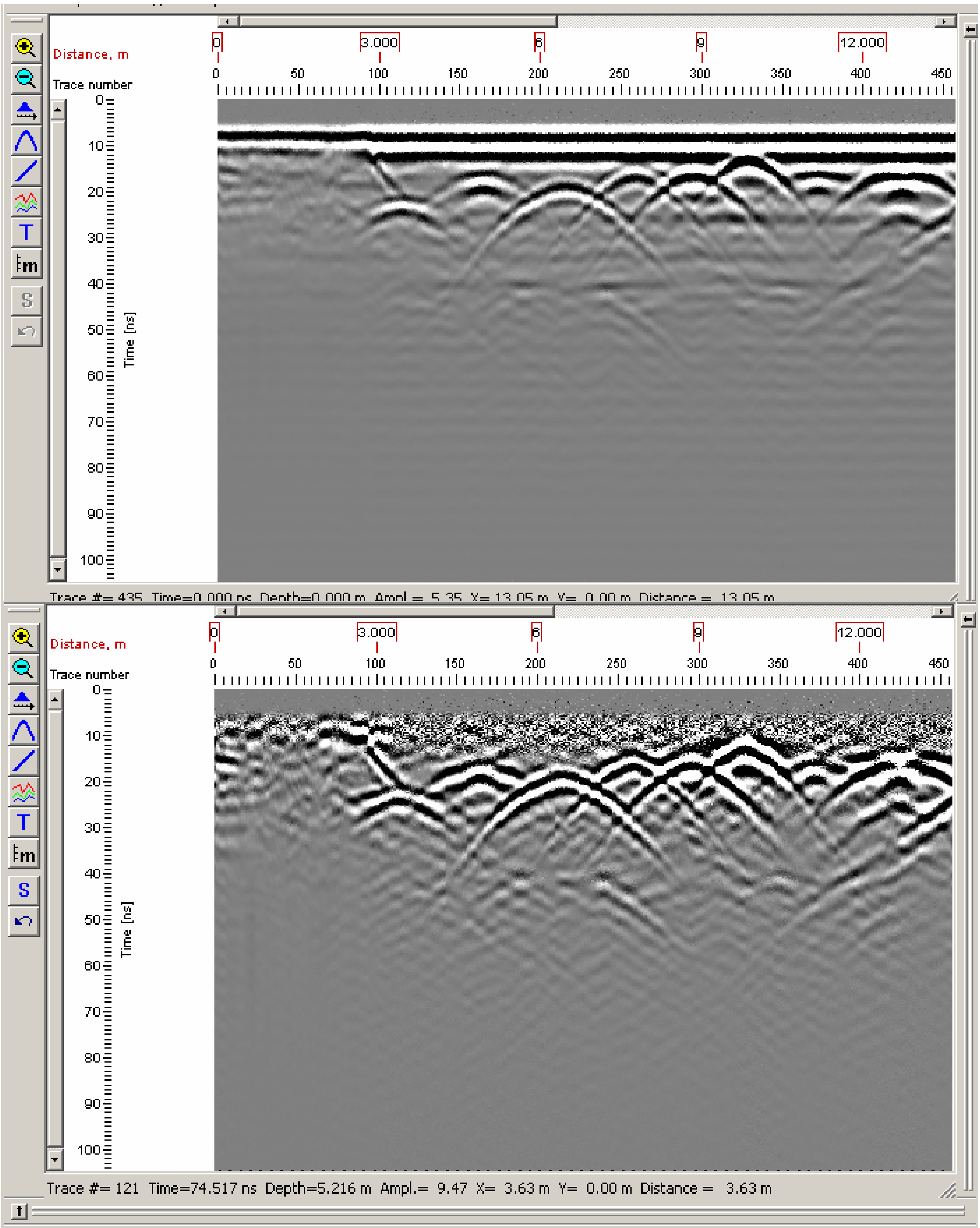



Fig. 5.5.5 – before and after the “*Background Removal*” routine application

***2D Spatial Filtering***

The "*2D Spatial Filtering*" module is designed to accomplish various types of spatial two-dimensional filtering.

When selecting this routine, in the "**Module parameters**" tab the following adjustment fields will appear (see Fig. 5.5.11):

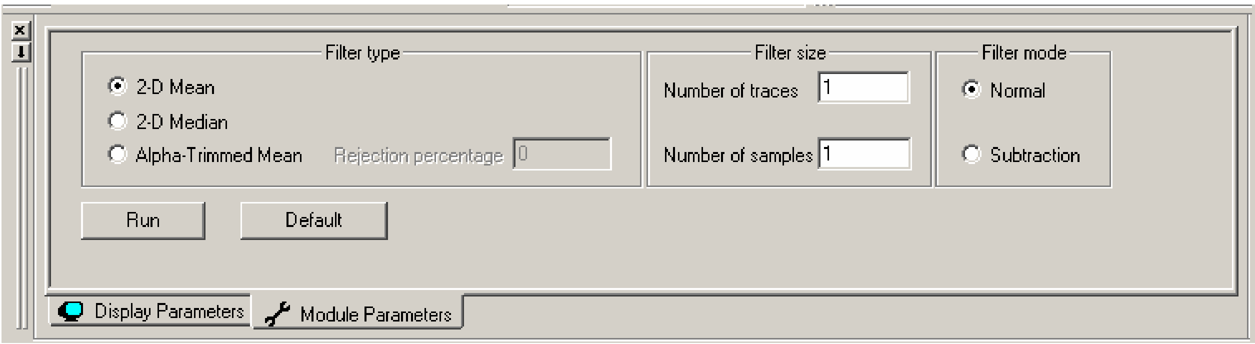



Fig. 5.5.11

Select the "Filter type":

- "2-D Mean" – this algorithm averages the samples within the bounds of filter application window;
- "2-D Median" – this algorithm sorts the samples within the bounds of filter application window and generates median (central) sample of the sorted set.
- "Alfa trimmed" – this algorithm sorts the samples within the bounds of filter application window and averages the range of values centralized regarding to the median. This process is equivalent to the process of culling of samples which are beyond the bounds and smoothing the rest of them.

The "Rejection percentage" filed becomes available if the "Alfa trimmed" filter is selected. In this filed one should specify the percentage of input samples of the filter window which will be discarded before averaging the rest of the samples. The 40% value means that the lowest 20% of the samples and the highest 20% of the samples will be discarded while filter calculation. When the 0% value is entered, the filter at the output will give the mean sum whereas the 100% value will give the median filter.

In the "Filter size" field it is necessary to specify the filter application window size, i.e. the dimensions of its operator – width and height.

- Specify the "Number of traces" that will be used as the width of the operator of two-dimensional spatial filter.
- Specify the "Number of samples" that will be used as height of the operator.

Select the "Filter mode".

- In the "Normal" mode the value of the sample in the center of the two-dimensional filter operator is replaced by calculated value (see the example on Fig. 5.5.12).
- In the "Subtraction" mode in the center of 2-D filter the calculated value is subtracted from the input value, thus the "differential" data set remains. This mode can be used instead of "Background removal" routine and in many cases can provide better results (see the example on Fig. 5.5.13).

*Warning!:* Owing to the median filters nature, when using the median filter the accidental gaps in values of samples may occur and they are the result of shifts in family of samples taken place while the process of sorting. It is recommended that the median filter is followed by the bandpass filter with a wide bandpass in order to eliminate these calculation gaps.

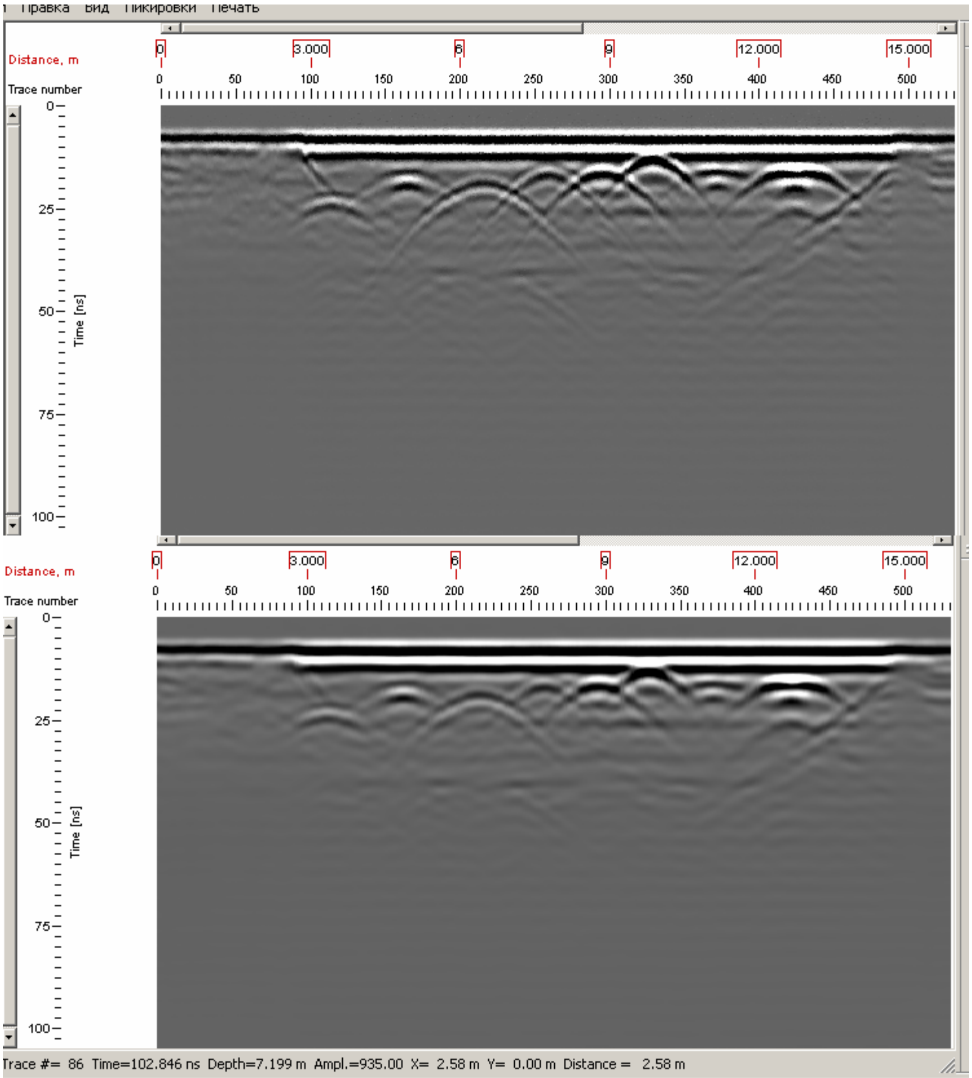


Fig. 5.5.12 – 2D filter application in a normal mode. Above – initial data, below – data after the “2D Filtering” routine application

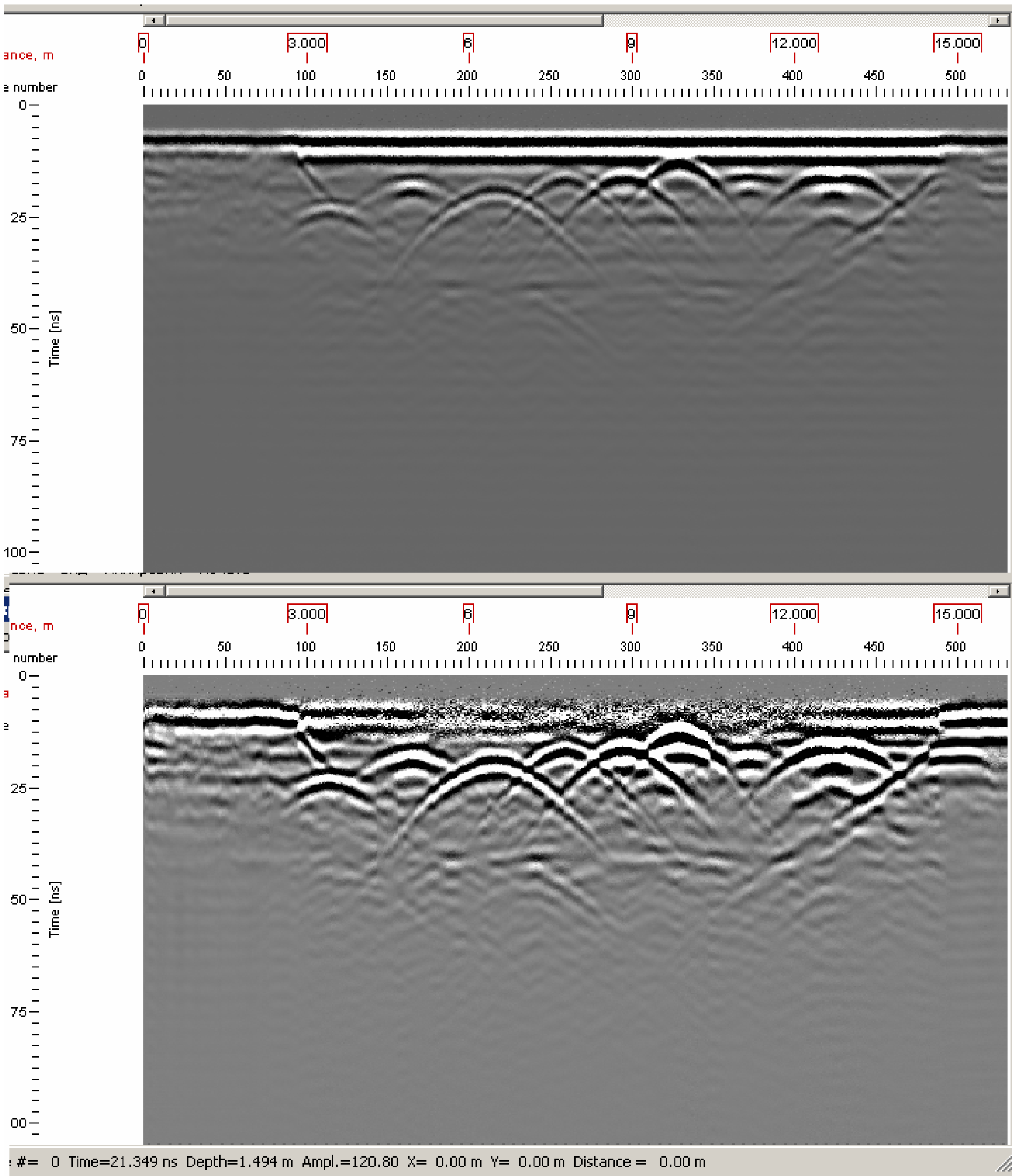


Fig. 5.5.13 – 2D filter application in a subtraction mode. Above – initial data, below – data after the "2D Filtering" routine application.

### *Amplitude Correction*

The "*Amplitude correction*" routine allows the user to accomplish gain control (amplitude correction) of different type. The module parameter window is shown on Fig. 5.5.14.

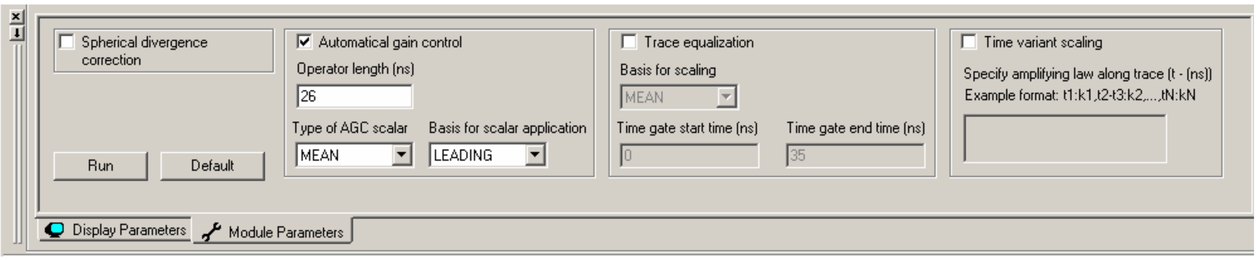



Fig. 5.5.14

The routine allows the user to accomplish 4 types of amplitude correction or any of their combination.

1. “Spherical divergence correction” applies to the traces the linear gain function that increases proportionally to the two-way travel time. The function compensates for amplitude losses at spherical divergence of wave front.

2. “Automatic gain control” automatically equalizes the amplitudes along the traces. The equalization of amplitudes occurs separately for every trace in the running window with specified length. The average amplitude is calculated for every position of the running window, after that the signal value in the application point (at the window beginning – by default) is divided by this average value. When selecting this routine one should set the following parameters:

- The “Operator length (ns)”. It is a value that defines a window length (expressed in ns) that will be used for gain calculation. The gain factor is calculated for every position of the window with specified length that slides down the trace with the one reading interval.
- The “Type of AGS scalar” is the type of automatic gain control coefficient:
  - “MEAN” – the mean of automatic gain control is calculated as arithmetical mean from absolute values of the amplitudes in the running window;
  - “RMS” – the mean of automatic gain control is calculated as a root-mean-square from absolute values of the amplitudes in the running window.

  To select a type of a coefficient one should click the left mouse button on the corresponding field of a drop-down menu;

- “Basis for scalar application” – is the position where automatic gain control coefficient is applied:
  - “CENTERED” – applies the coefficient to the central sample in every running window;
  - “LEADING” – applies the coefficient to the first sample in every running window;
  - “TRAILING” – applies the coefficient to the last sample in every running window.

  To select the position of coefficient application one should click the left mouse button on the corresponding field of a drop-down menu.

3. The “Trace equalization” is used for decreasing amplitude variations from trace to trace. For every trace within the bounds of specified time window the mean value of amplitudes is calculated and then all the trace values are divided by this mean value. Thus, every trace becomes normalized on its balancing coefficient and amplitude variations decrease from trace to trace. The parameters of this function are:

- “Basis for scaling” – defines the way the balancing coefficient of the trace is calculated:

    - "Mean" – every trace will be normalized according to the mean absolute value of the amplitudes in a specified analysis window;
    - "RMS" – every trace will be normalized according to the root-mean-square value of the amplitudes in a specified analysis window;
    - "Maximum" – every trace will be normalized according to maximum absolute value of the amplitudes in a specified analysis window.

- "Time gate start time (ns)" – the initial time of the analysis window.
- "Time gate end time (ns)" – the finite time of the analysis window.

4. The "Time variant scaling" option allows manual specification of arbitrary law of gain variation along the trace. Having selected this routine one should set the pair "time-gain" in the opened window. The order of specification is the following:

*time t1 : coefficient K1, time t2 – time t3 : coefficient k2, …, time tN : coefficient kN*

Between the specified values the gain factor will interpolate linearly.

The example of the "Amplitude correction" routine application in the "Automatic gain control" mode, where the parameters by default are used is shown on Fig. 5.5.15.

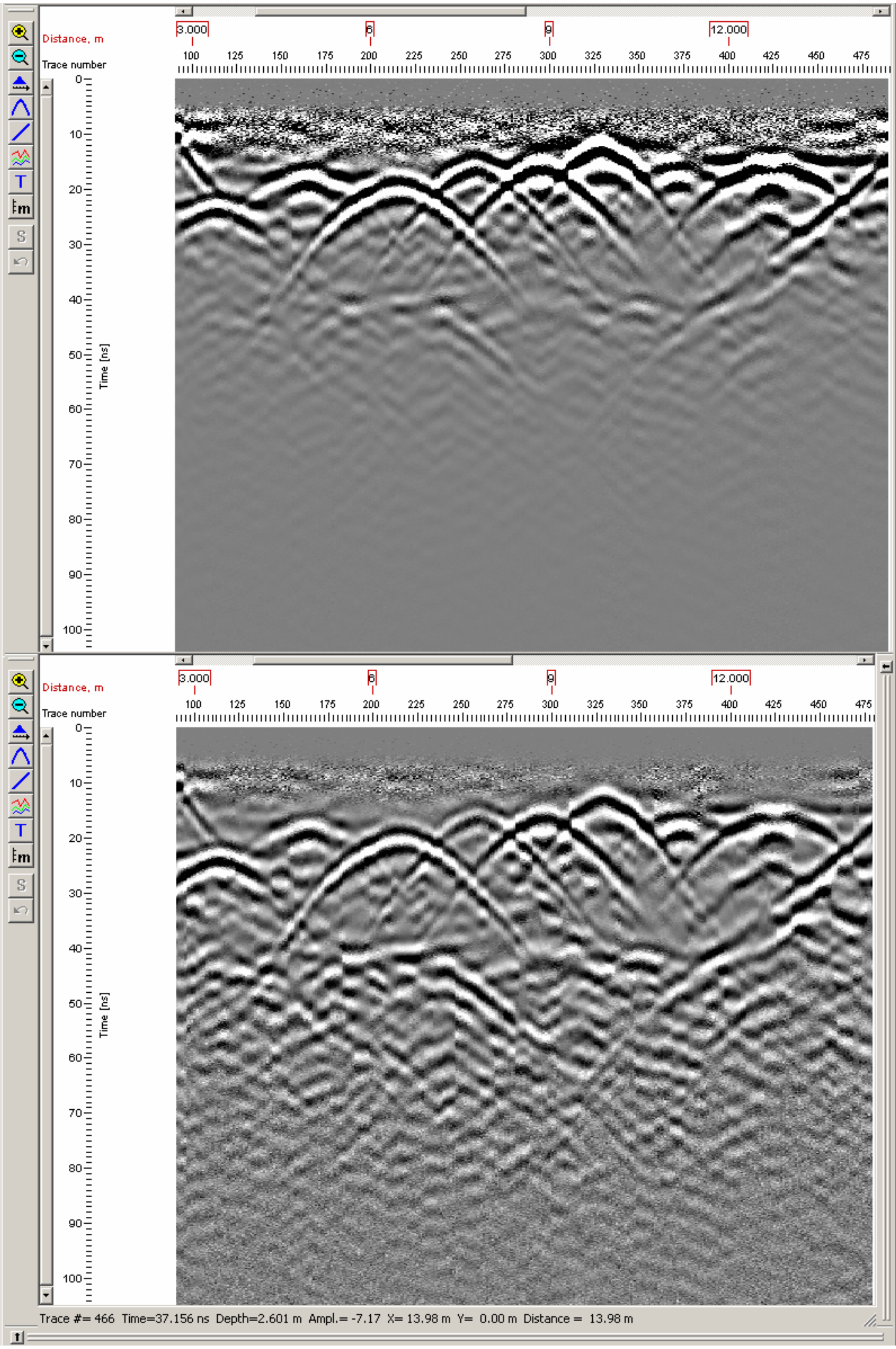



Fig. 5.5.15 – the data before and after the “Amplitude correction” routine application in the “Automatic gain control” mode

### ***Predictive Deconvolution***

The "*Predictive Deconvolution*" routine is applied in order to increase the resolution due to "narrowing" of reflected wavelets. In this case, as a rule, the undesirable increases of the noise level take place and that is why almost always right after deconvolution the *Bandpass filtering* should be applied. More details concerning the predictive deconvolution algorithm one can find in the book *Hatton, L., Worthington, M.H., Makin, J., 1986. Seismic data processing: theory and practice. Oxford: Blackwell Scientific.*

The routine parameters window is shown on Fig. 5.5.16:

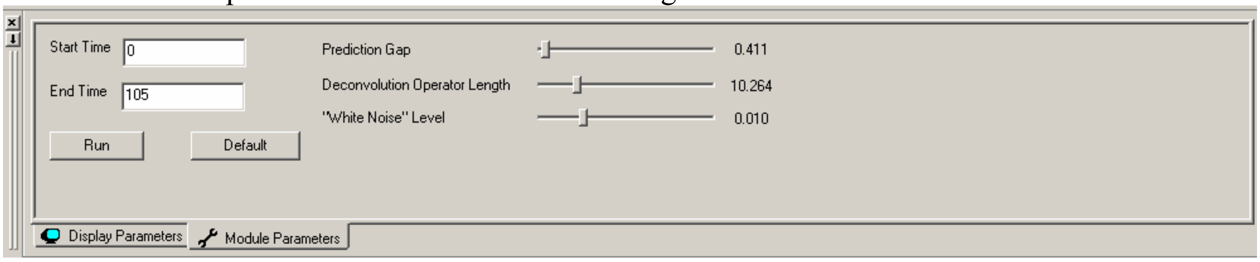



Fig. 5.5.16

In the "Start time" and the "End time" fields one should specify the boundaries for analysis window (expressed in nanoseconds) according to which the deconvolution operator will be computed.

The other parameters of the routine one can adjust with the help of the sliders.

The "Prediction gap" defines the compression rate of the wavelets. The lower values of the parameter correspond to shorter resultant wavelet and, therefore, to higher resolution. At the same time one should always keep in mind that the higher the compression rate is (i.e. the shorter deconvolution interval is) the more apparent the noise level increasing effect becomes.

The "Deconvolution Operator Length" is a value of maximum autocorrelation delay. It defines the range of deconvolution operator operation. At low parameter values, after routine application there is only wavelet compression, at high values – one may also observe the decay of clutters (multiple reflections, antenna "ringdown", etc.)

The ""White Noise" level" is a stabilizing parameter. The higher the parameter level is the lower the undesired effect of noise level increasing after routine application. This parameter in not very important and plays a noticeable role only at short prediction gaps.

The example of the "*Predictive deconvolution*" application is shown on Fig. 5.5.17.

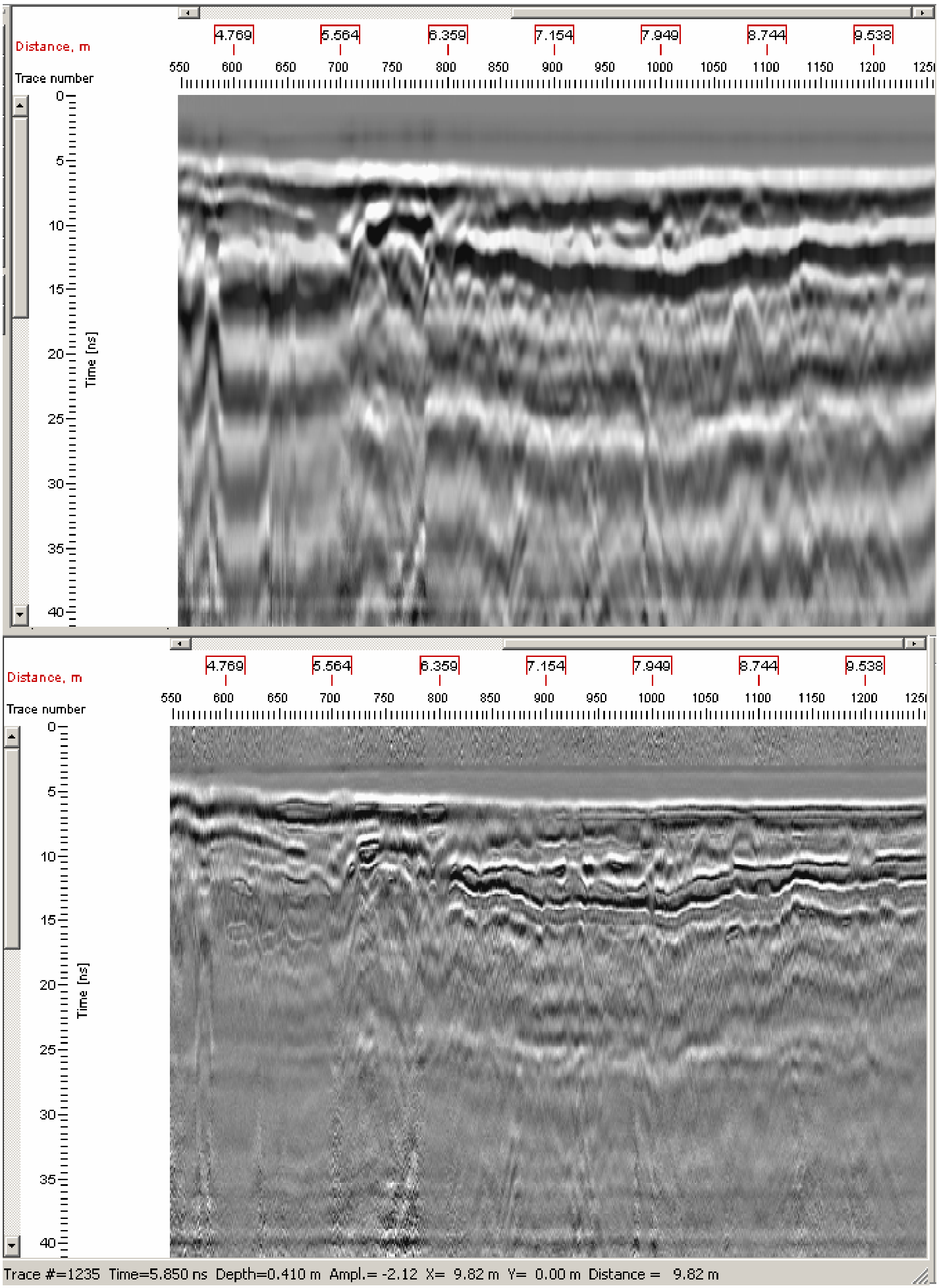
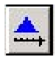



Fig. 5.5.17 – the data before an after the "*Predictive Deconvolution*" routine application

## *Bandpass Filtering*

The "*Bandpass Filtering*" routine is used to increase the signal/noise ratio. The filtering algorithm operates within frequency domain and is realized by means of multiplication of trace Fourier transform (its frequency spectrum) on the spectrum (amplitude-frequency response) of the filter. A simple trapeziform zero-phase bandpass filter is used in this programme. The amplitude-frequency response of the filter that will be applied to data is displayed in the dialog box of the routine parameters (see Fig. 5.5.18) (To view the spectrum of data use the  "Spectrum" tool on the toolbar)

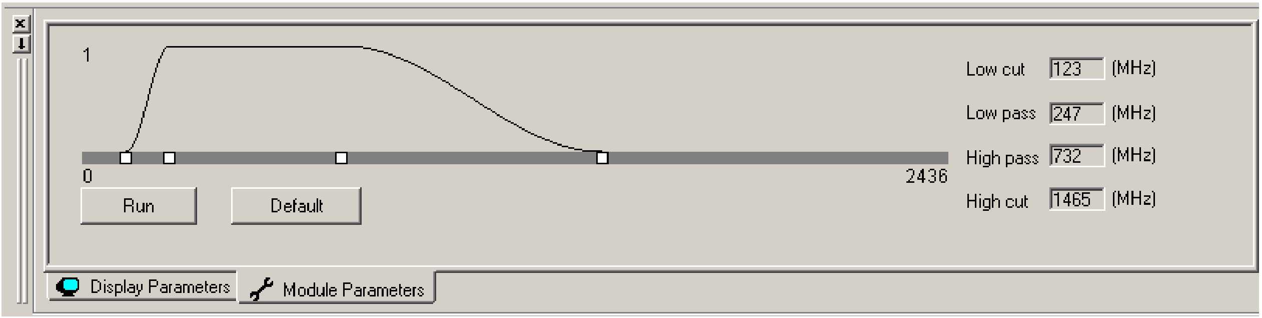


Fig. 5.5.18

The setup window is conditionally divided into two parts: in the left part – graphical presentation of a filter form, in the right part – numerical value of its parameters: "Low cut", "Low pass", "High pass" and "High cut".

These values consistently define the 0% and 100% points of signal gating from the side of lower frequencies and 100% and 0% points of signal gating from the side of higher frequencies (expressed in MHz). The filter slopes develop within a frequency domain of linear weight function.

In order to set any of four parameters one should press the left mouse button on a graphic element of the corresponding parameter and holding it pressed move it to the required position on a frequency scale. The parameter value will be displayed in the corresponding field in the right-hand part of the window.

Below one can fine an example of the routine application to the data after predictive deconvolution (see Fig. 5.5.19). In the right bottom angle of visualization windows there are data spectra (amplitude-frequency characteristics) before and after bandpass filtering respectively.

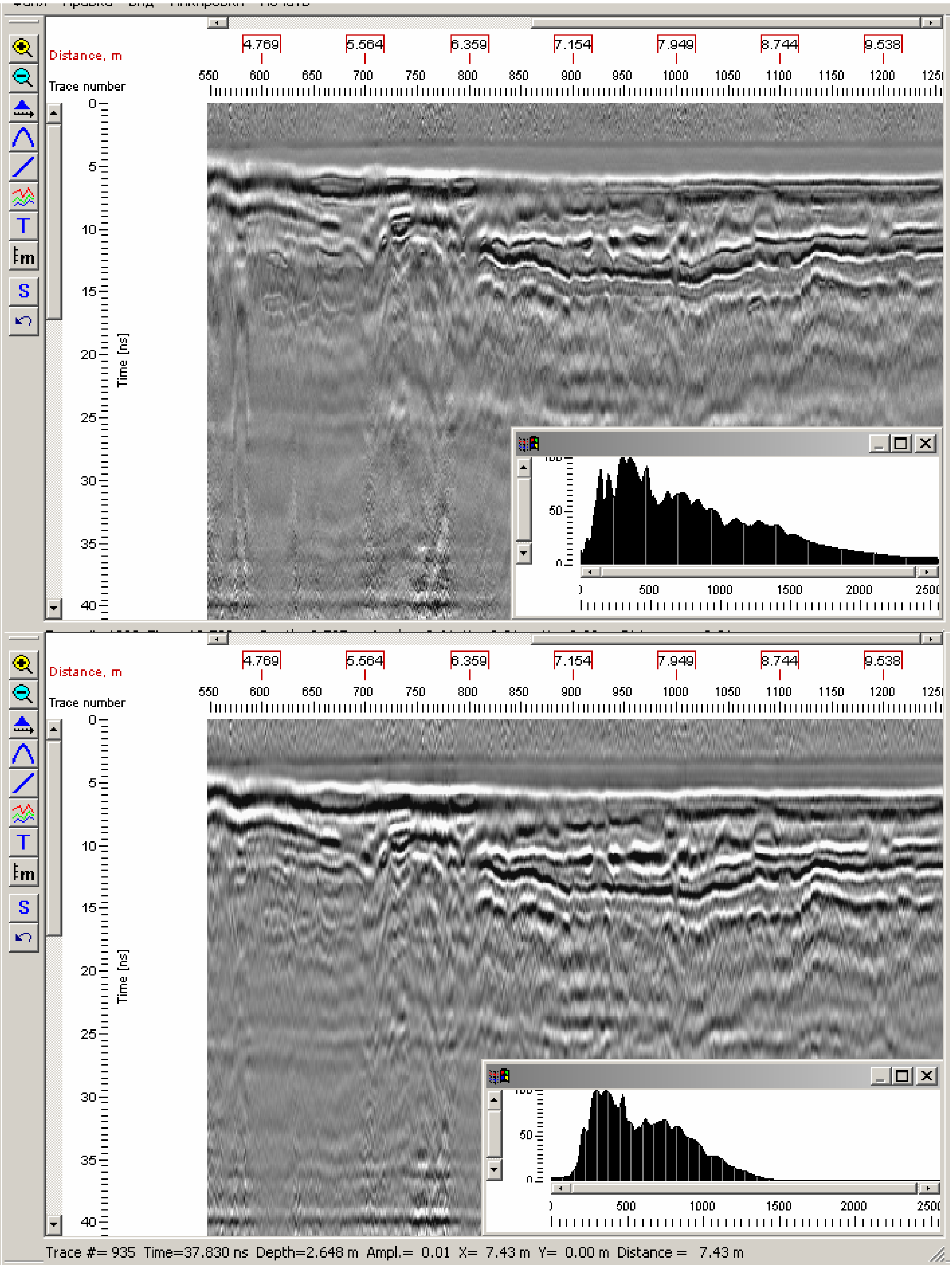


Fig. 5.5.19 – the radargram and its spectrum before an after the “*Bandpass Filtering*” routine application

### *Stolt F-K Migration*

The “Stolt F-K Migration” routine is applied in order to restore the real location and shape of reflecting boundaries in a section plane. Particularly, if the routine has been applied correctly the hyperbolic reflections (the diffracted waves) gather in points which locations correspond to diffracting local objects.

The migration algorithm used in the programme (Stolt migration algorithm in F-K domain), as compared with other migration algorithms, is very fast. The main restriction of this algorithm is that the wave propagation velocity within the limits of the section fragment that is under processing considers being constant.

For more details about the migration algorithm one can address to the book *Hatton, L., Worthington, M.H., Makin, J., 1986. Seismic data processing: theory and practice. Oxford: Blackwell Scientific.*

The only parameter for this routine (Fig. 5.5.20) is an electromagnetic wave propagation velocity within the medium – migration velocity. Specify the “Migration velocity” with the help of a slider in the dialog box of routine parameters window.

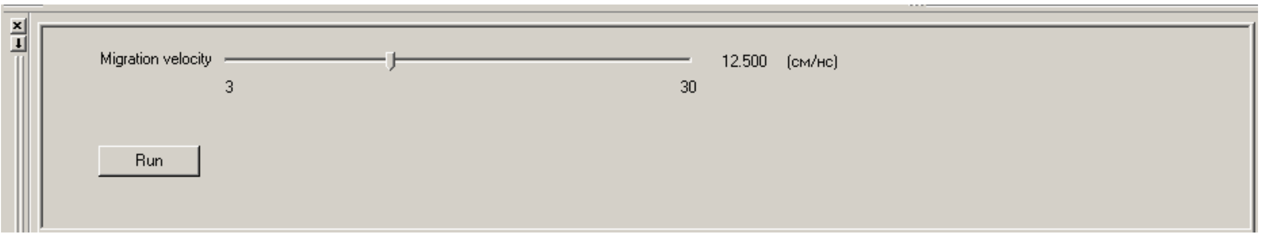


Fig. 5.5.20

If a velocity value is specified incorrectly the obtained result can be even farther from the true one than the initial result (“overmigrated” section when velocity is overstated and the “undermigrated” section when velocity is understated). In order to determine migration velocity one can use the “Hyperbola” tool ( “Hyperbola”) on the toolbar.

The example of migration application is shown on Fig. 5.5.21.

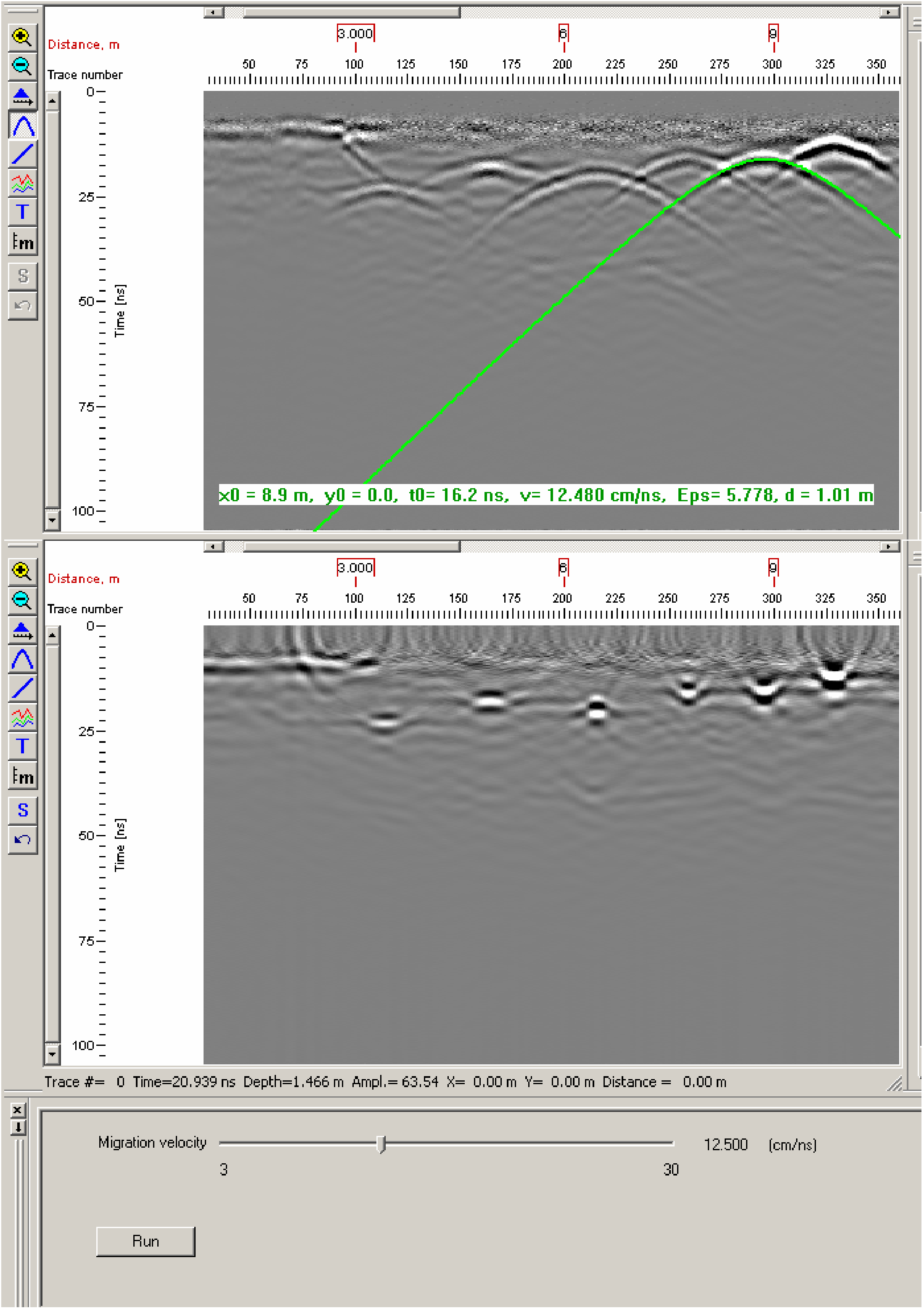



Fig. 5.5.21 – the radargram before an after the "*Stolt F-K Migration*" routine application. The migration velocity has been determined with the help "Hyperbola" tool.

### *Reflection Strength*

The "*Reflection Strength*" routine is used for radar trace converting into instantaneous amplitude ("reflection strength"). The reflection strength calculation is accomplished with the help of the Hilbert transform. As a matter of fact, the reflection strength trace can be considered as an envelope of the initial trace.

The radargram visualization as instantaneous amplitudes often allows the user to more accurately trace the amplitude variations along the reflecting boundaries and in the whole section. It is sometimes useful when there is a necessity to select and to emphasize on the radargram the areas with representative record type which is different from the adjoining areas in higher and lower amplitude of the signal.

When this routine has been applied the data cannot be further processed. If one needs to represent the radargram as a section of instantaneous amplitudes one should use this routine at the very end of the processing. After the "*Reflection Strength*" routine has been applied it is possible, if required, just to smooth the obtained image with the help of "*2D Spatial Filtering*" in the normal mode and to make topography correction with the help of "*Topography*" routine.

When invoking the routine, on a second tab of the parameters panel the following dialog box will appear (see Fig. 5.5.22).

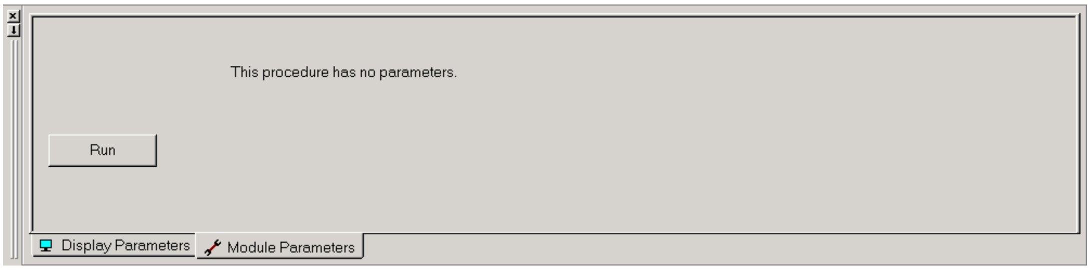



Fig. 5.5.22

This routine does not have any parameters.

The example of application of this routine is presented on Fig. 5.5.23 (the initial data) and on Fig. 5.5.24 (after transformation).

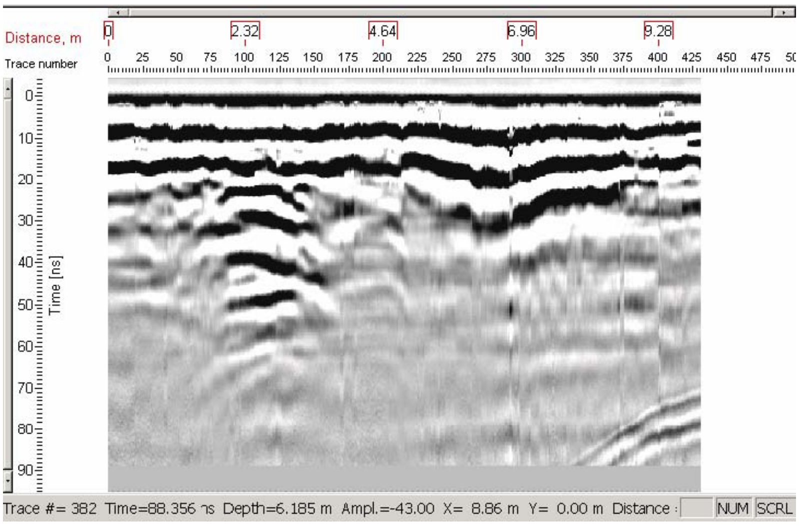



Fig. 5.5.23

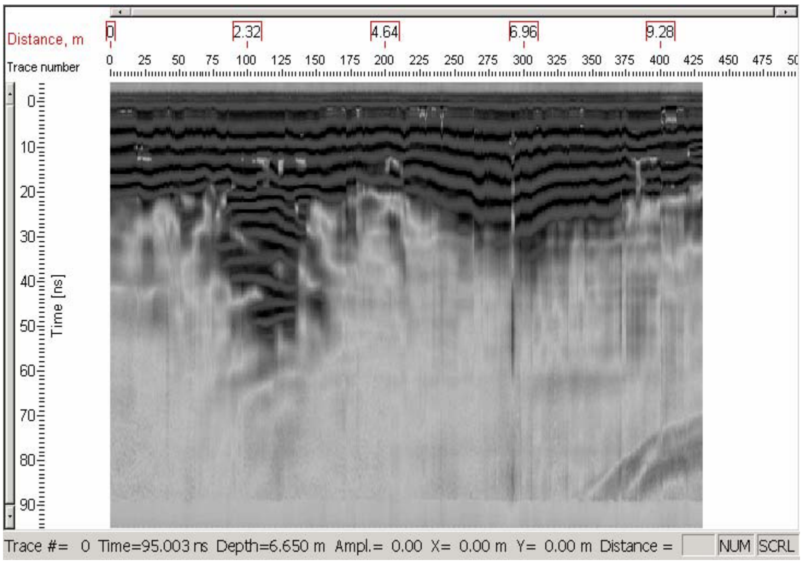



Fig. 5.5.24

### *Topography*

The *Topography* routine is meant for correcting the data for the surface topography along the profile.

Topography is expressed in meters or feet. One can specify the topography expressed in absolute or relative values – the programme converts the topography values into static corrections with regard to the highest point. Between the specified points and beyond them the topography interpolates linearly.

The topography corrections are entered into the data represented in time scale. It means that in addition to topography specification in meters or feet, it is also necessary to specify the electromagnetic wave propagation velocity within the medium, which is required for topography converting into static corrections expressed in nanoseconds.

The window for routine parameters specification is shown on Fig. 5.5.25.

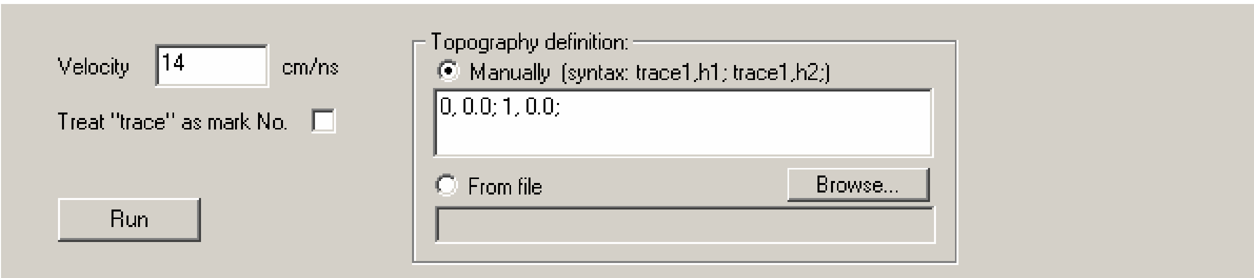



Fig. 5.5.25

In the “**Topography definition**” field one can chose the manual topography selection mode or from a space separated tabular text file. When the manual topography selection mode is chosen one should specify the height value (expressed in meters or feet) in the known points according to the following sample:

*trace1_number, height1; trace2_number, height2;… traceN_number, heightN;*

When the “**From file**” option is selected one should press the “**Browse**” button in order to select a space separated tabular text file with topography description. The file should be in the following format:

*trace1_number height1*
*trace2_number height2*
*…*
*traceN_number heightN;*

In the "**Velocity**" field one should specify the wave propagation velocity within the medium required for topography converting into static corrections for traces.

If one activates the "**Treat "trace" as marker number** option the *trace_number* values will be taken as the numbers of reference marks made by GPR operator in field. (If the option is activated but any of the specified mark are lacking in the file then this number will be taken as the trace number).

It is not recommended to apply other processing routines after the *Topography* routine has been applied. This routine should be used at the final stage of data processing.

The example of *Topography* routine application is presented on Fig. 5.5.26.

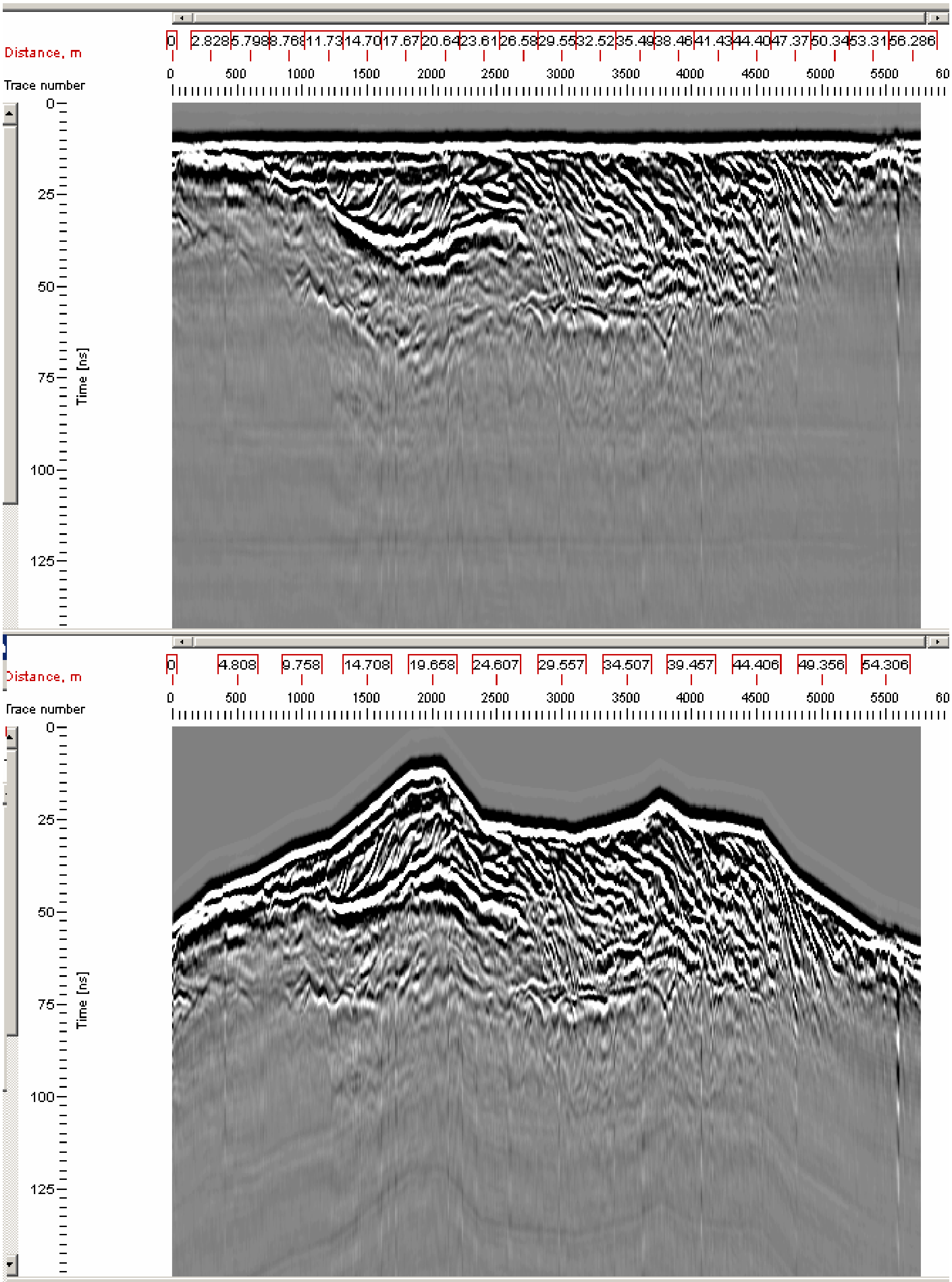


Fig. 5.5.26 – the radargram before and after topography correction

## “Flow” tab

All the processing routines applied to the data are automatically included into processing history or “flow”. To do operations with the flow there is a “**Flow**” tab on the routines panel (see Fig. 5.5.27).

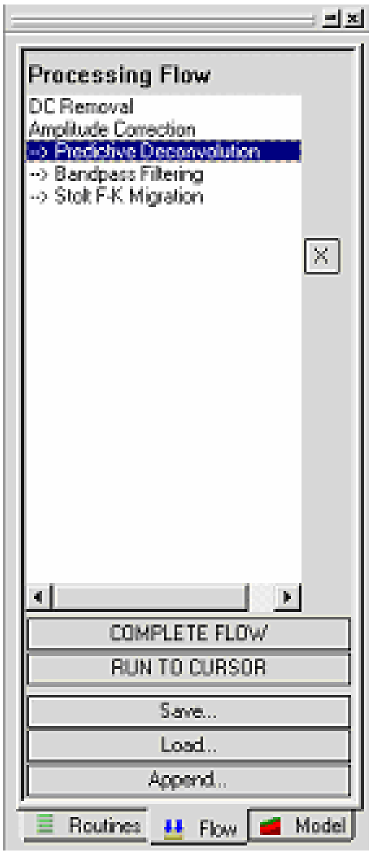


Fig. 5.5.27

When working with this tab you can undo the applied routines, apply them again but with different parameters, delete the routines from the flow. The flows can be saved as files on disk, loaded from the disk and reused with other files.

The flow is filled in top-down, i.e. the first applied processing routine will be displayed at the top of the flow, and the last routine that has been applied will be displayed at the bottom.

The routines in the flow can be in applied or canceled (not applied) state. In the flow the canceled (not applied) routines are marked by the arrow before the routine name (on Fig. 5.5.27 the *Predictive Deconvolution, Bandpass Filtering* and *Stolt F-K Migration* routines have been canceled).

The "Flow" tab contains the following elements:

- "Processing flow" contains the list of routines in the flow (both applied routines and not applied ones);
- The button allows the user to delete a routine from the flow;
- The "Complete flow" button allows the user to execute all not applied routines until the end to the flow;
- The "Run to cursor" button allows the user to execute all not applied routines of the flow until and including the selected one. The routines below the selected one will not be executed;
- The "Save …" button allows to save the flow into a disk-file;
- The "Load …" button allows loading the flow from a disk-file. In this case all the processing applied before will be canceled and the active flow will be replaced by the flow from the selected file. All the routines of the flow will appear in the not applied state;

- The "Append …" button allows adding the flow that has been saved before to the active one. Routines of the flow from selected disk-file the will be added to the end of the active flow in the not applied state.

***The main commands to do operations with routines in the flow***

*Undo*

To undo the routine in the flow, single-click left mouse button on it. By doing this, all the following routines of the flow will be canceled and the settings for the selected (last canceled) routine will open in the "**Module parameters**" tab of the parameters panel.

*Changing parameters of routines in the flow*

After the routine in the flow has been canceled one can change its parameters. To do this one should click the left mouse button on one of the canceled routines in the flow. Its parameters will open in the "**Module parameters**" tab of parameters panel. Now you can change the parameters of the routine. If you decide to apply the routine again now, it will execute but with new parameters.

*Redo of canceled routines*

To apply the routine that is in the canceled state, one should double-click the left mouse button on it or press the "**Run**" button in the window with parameters settings for this routine. At this, all the canceled routines prior to the selected one and the selected routine will be executed. The same result can be achieved by selecting the routine in the flow by the left mouse button and pressing the "**Run to cursor**" button on the "Flow" tab.

*Deleting routine from the flow*

After the routine in the flow has been canceled, one can delete it from the flow. With the help of the mouse select the routine that you desire to delete and press the  button.

To save the flows on disk and to load them again use the "**Save …**", "**Load …**" and **"Append…"** buttons.

*NOTE:* The possibility to re-use the flows is especially convenient while processing the data bulk obtained with the same recording parameters (for example, several profiles of a single survey). In case when you have created the flow working with one kind of data and then you decide to apply them to the data of different kind (different record length, different sampling frequency) it may appear that the parameters of the routines in the saved flow are not compatible with the new data. In this case you may either change the parameters of the routine in the flow manually or create a new flow.

##  "Model" Tab

The **RadExplorer** programme allows the user not only to accomplish digital signal processing but also to create velocity models of the sections and this allows execution of one of the most important operations of data interpretation – converting data from time section to depth section.

Activation of velocity model editor allowing layers creation and editing, as well as time to depth conversion is performed in the "**Model**" tab (Fig. 5.5.28).

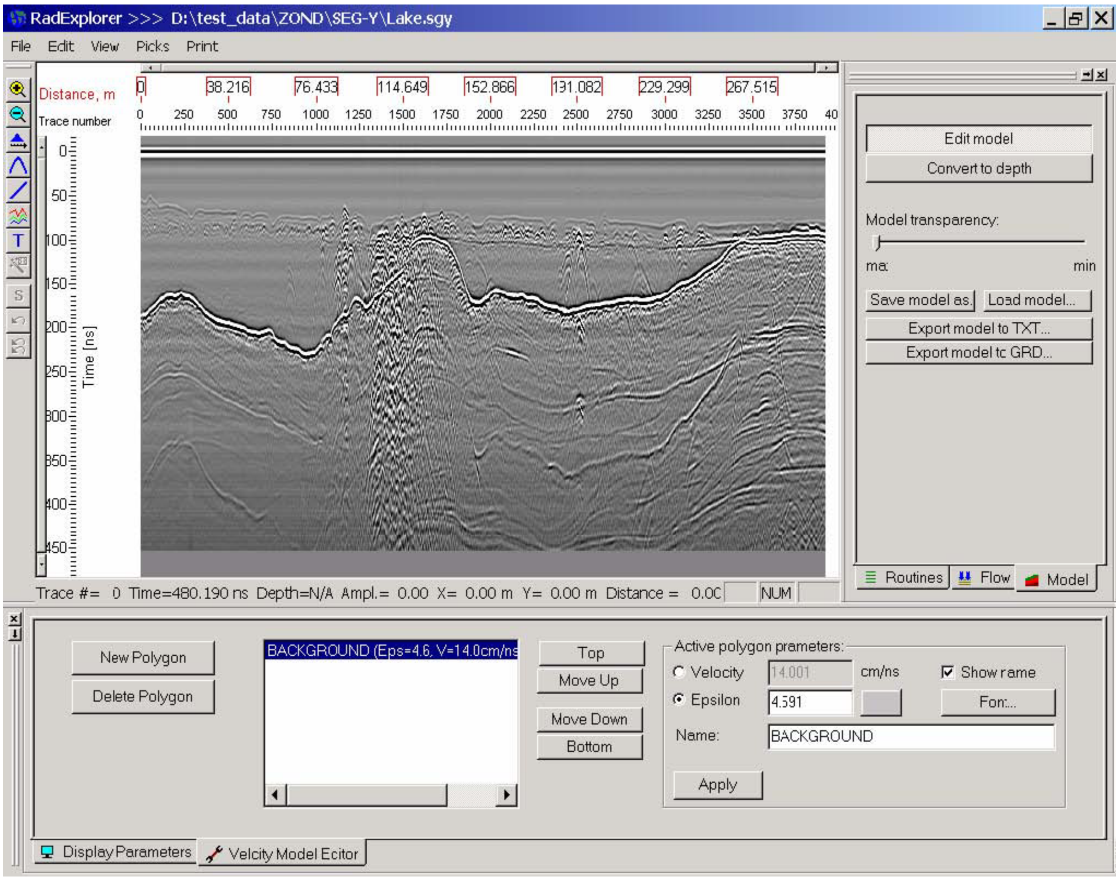



Fig. 5.5.28

When one opens this tab, the “Velocity model editor” tab providing interface for the model creation and editing will appear in the Processing parameters panels instead of the “Module parameters” tab.

At first let us consider the features of “**Model**” tab (Fig. 5.5.29).


Fig. 5.5.29

It contains the following elements:

- “Edit model” button enables the model editing mode;
- “Convert to depth” button enables converting the time section to depth section in accordance with the current velocity model (since the data presented in depth scale cannot be processed, then when one switches to the “Routine” or “Flow” tab, the radargram automatically returns to the time scale);
- “Model transparency” field with a slider sets the transparency of the model. At maximum transparency only the radargram can be viewed and at minimal transparency – only the model. Before you start working with the model editor, move the “Transparency” slider a to and back to achieve the model transparency level most convenient for you;
- “Save model as …” button allows the user to save the model into file with *.vm extension;
- “Load model” button allows model loading from a file with *.vm extension;
- “Export model to TXT …” button performs model exporting into text file;
- “Export model to GRD …” button performs model exporting into file with *.grd extension (standard grid-file of Surfer programme).

The model in the **RadExplorer** programme is a set of polygons with specified values of dielectric constant/electromagnetic wave velocity.

When opening a data file, the model containing only one polygon (BACKGROUND) is created by default. The background occupies the whole data area and is colored in gray by default, its dielectric constant value is set as equal to the value that has been specified by GPR operator while recording (if this kind of information do not exist the value is set by default).

## Model Editor

Editing of already existing layers of the model and creation of new ones can be done with the help of "**Velocity model editor**" tab on the parameters panel (Fig. 5.5.30)

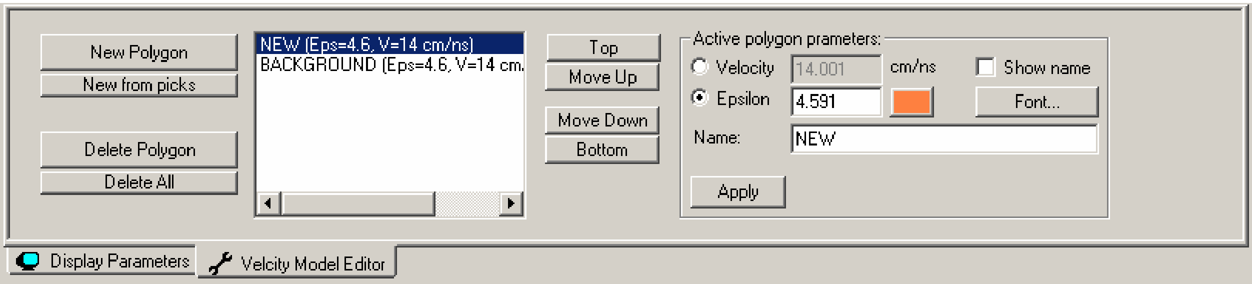



Fig. 5.5.30

"**Velocity model editor**" tab interface contains the following elements:

- "**New polygon**" button allows user to create a new polygon;
- "**New from picks**" button allows creating a new polygon from 2 existing horizon picks;
- "**Delete polygon**" button allows the user to delete selected polygon;
- "**Delete all**" button allows deletion of all polygons of the model (except BACKGROUND);
- A field of polygon list with their properties;

To the right of the list there are buttons that assign the order of polygons position with regard to one another:

- "**Top**" button allows the user to relocate the selected polygon to the top position;
- "**Move up**" button allows the user to relocate the selected polygon one position uwards;
- "**Move down**" button allows the user to relocate the selected polygon one position downwards;
- "**Bottom**" button allows the user to relocate the selected polygon to the bottom position, right above the BACKGROUND;

In the "**Active polygon parameters**" field one can specify the following parameters to the selected polygon (i.e. a layer or a body in the model):

- "**Velocity**" – wave propagation velocity expressed;
- "**Epsilon**" – dielectric constant;
- "**Name**" – the layer name;
- "**Show name**" option allows monitoring the polygon name displaying on the screen (applied to all polygons);
- "" button allows to chose the polygon filling color;
- "**Font …**" allows to chose the font for the polygon label string;
- "**Apply …**" apply the selected parameters to the current polygon.

To create a new polygon one should press the "**New polygon**" button. In the filed with list of layers (polygons). The new string called NEW with the same properties as the BACKGROUND will appear.

***The main commands for polygon editing:***

- *Add point* is performed by pressing the left mouse button. The point will appear on the screen at cursor position;
- *Relocate point* is performed by the right mouse button. Capture the point you would like to relocate by the right mouse button and, holding it pressed, drag it to the required place;
- *Delete point* is accomplished by double-clicking the right (additional) mouse button;
- *Move of the polygon as a whole* is accomplished by dragging it by the right mouse button and pressing the Shift key simultaneously.

One can create as many layers as it is necessary. The order of their disposition in the list should be specified with the help of "**Top**", "**Move up**", "**Move down**", "**Bottom**" keys. At this, the BACKGROUND layer (base plate) will always remain the bottommost layer (at the same time you can always change its name and properties). When converting into depth or exporting model, the velocity/dielectric constant in one or another point of a section is taken in accordance with the parameters of the uppermost polygon in this area, i.e. the polygon that is displayed on the screen.

The visualization and velocity parameters of selected polygon should be specified in "**Active polygon parameters**" field. In order to make the polygon active you should select it from the list of created layers by clicking the left mouse button.

Use **New from picks** button to create a new model polygon based on two existing picks. A dialog box with a list of available picks will appear (Fig. 5.5.31).

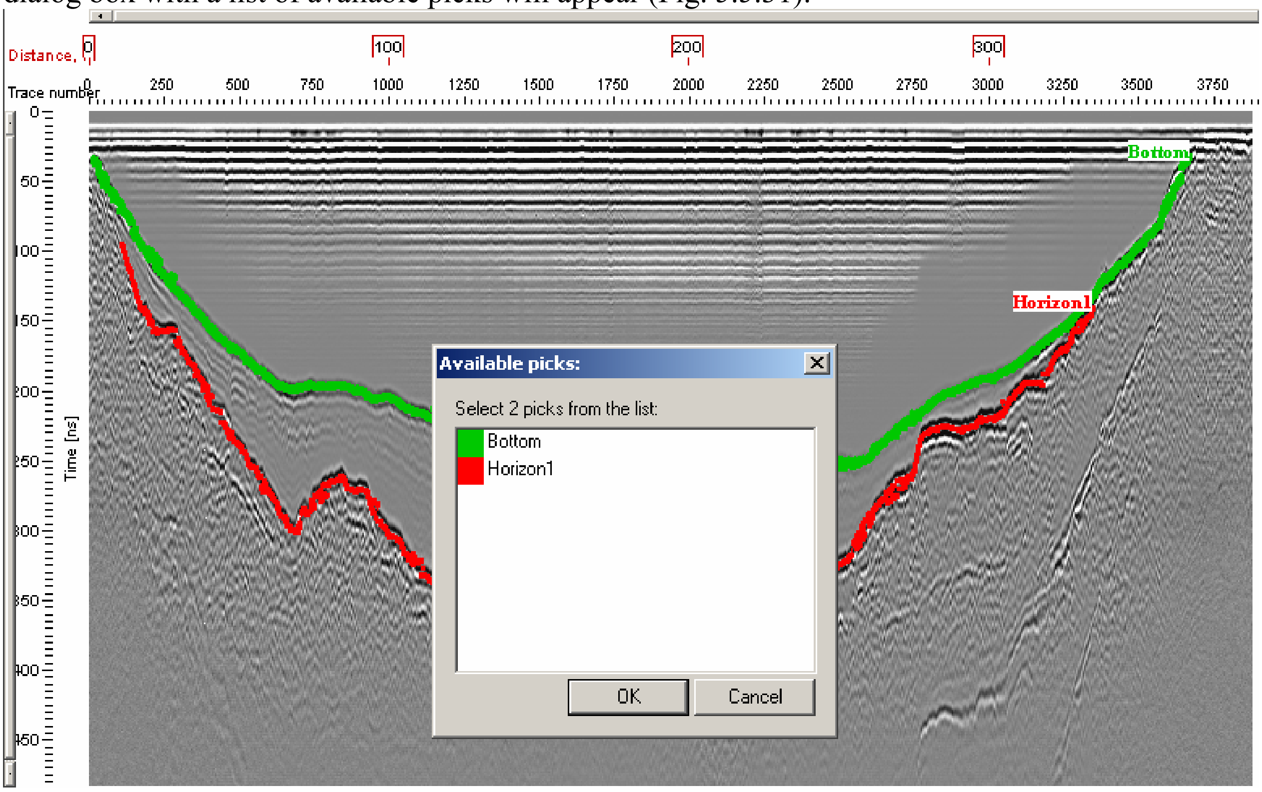



Fig. 5.5.31.

Select two picks in the list (use Ctrl key when selecting the second one) and press **OK**. A new model polygon will be created basing on the overlapping portions of the selected picks (Fig. 5.3.32).

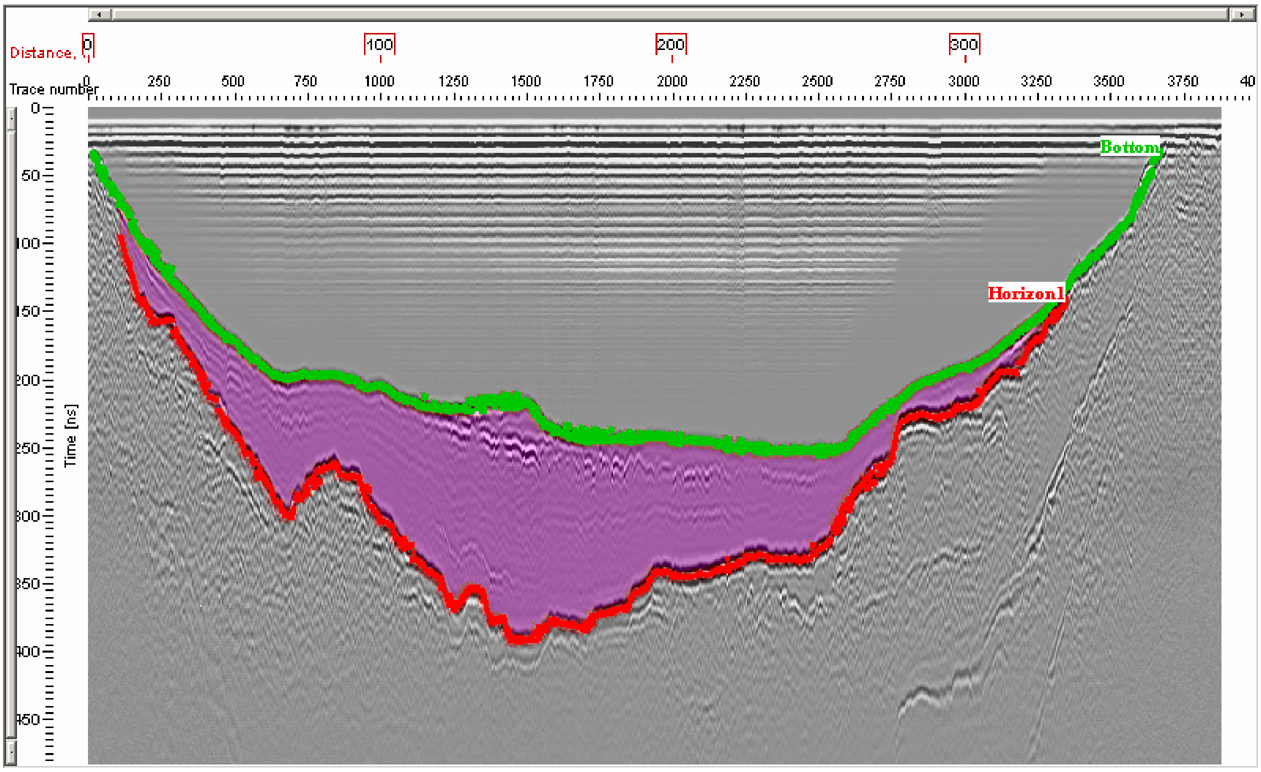



Fig. 5.3.32.

The examples of initial data file (see Fig. 5.5.33), velocity model created for it (see Fig. 5.5.34) and of the section converted to depth (see Fig. 5.5.35) are shown below.

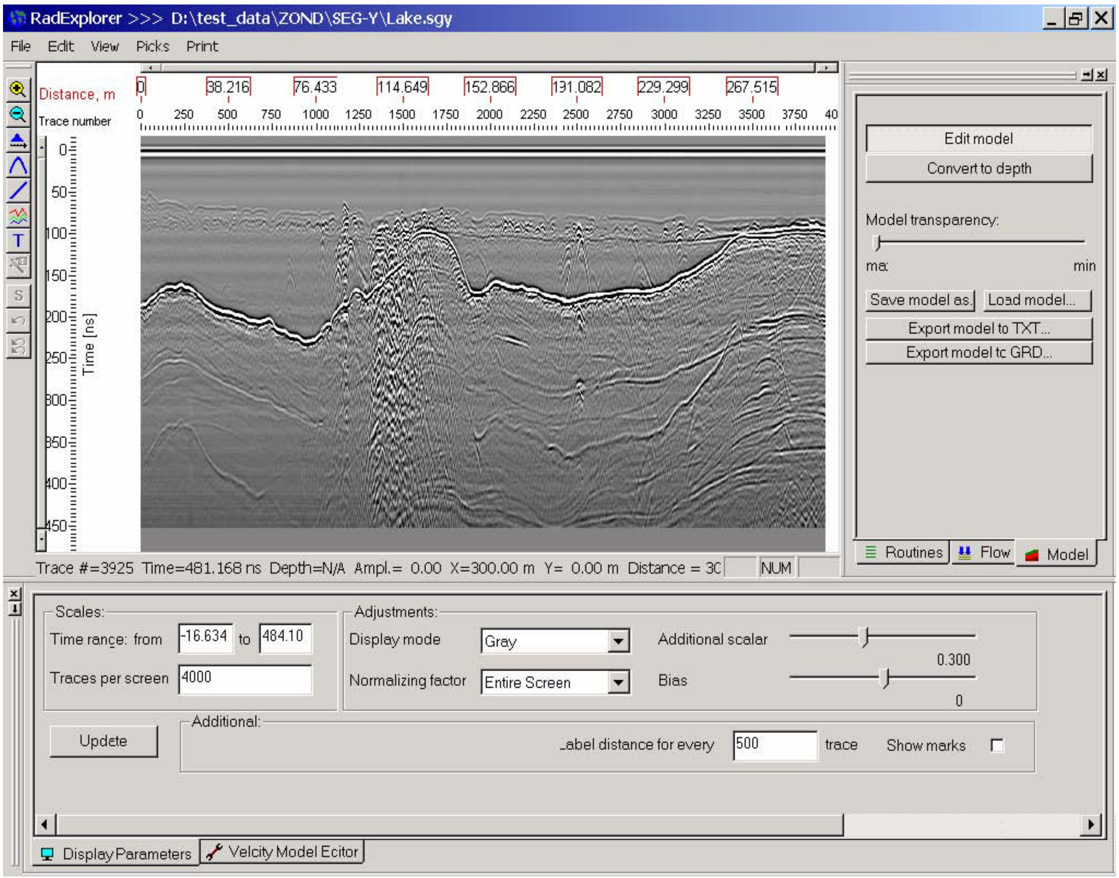



Fig. 5.5.33 – the initial file in a time scale

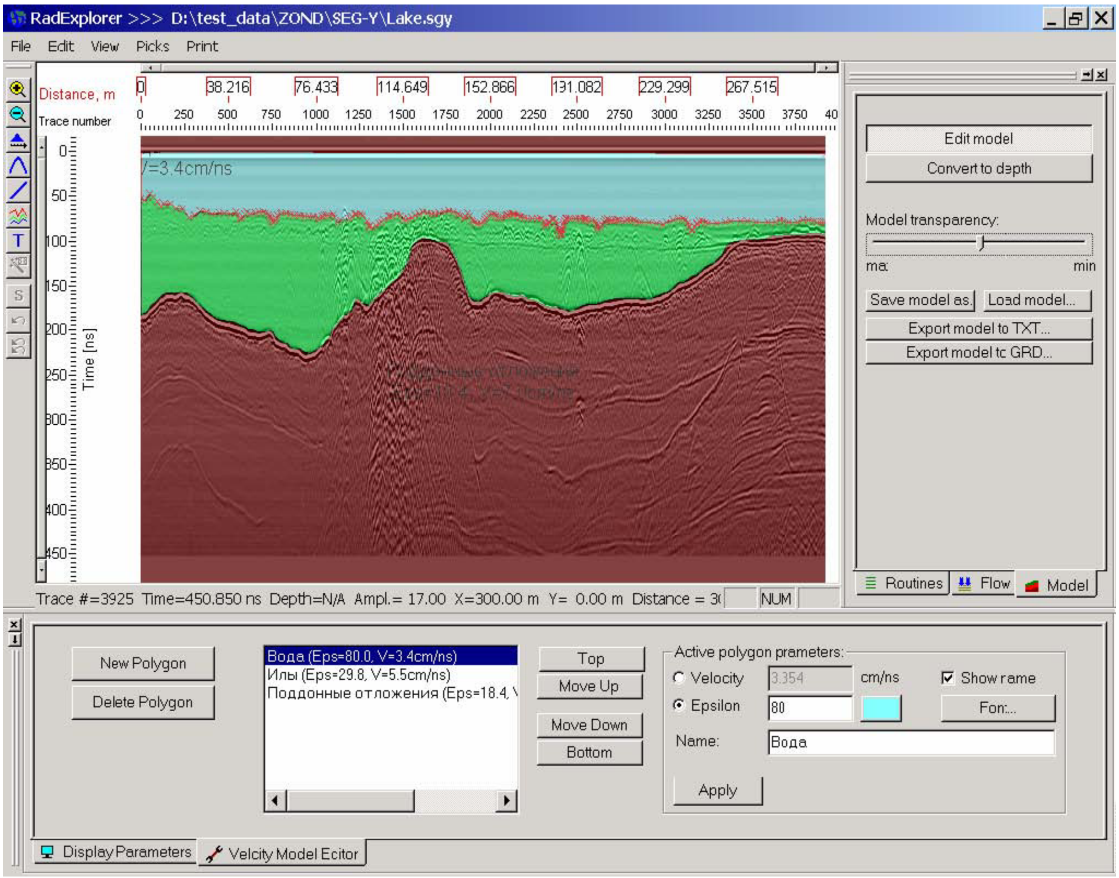



Fig. 5.5.34 – the radargram and the model created for it

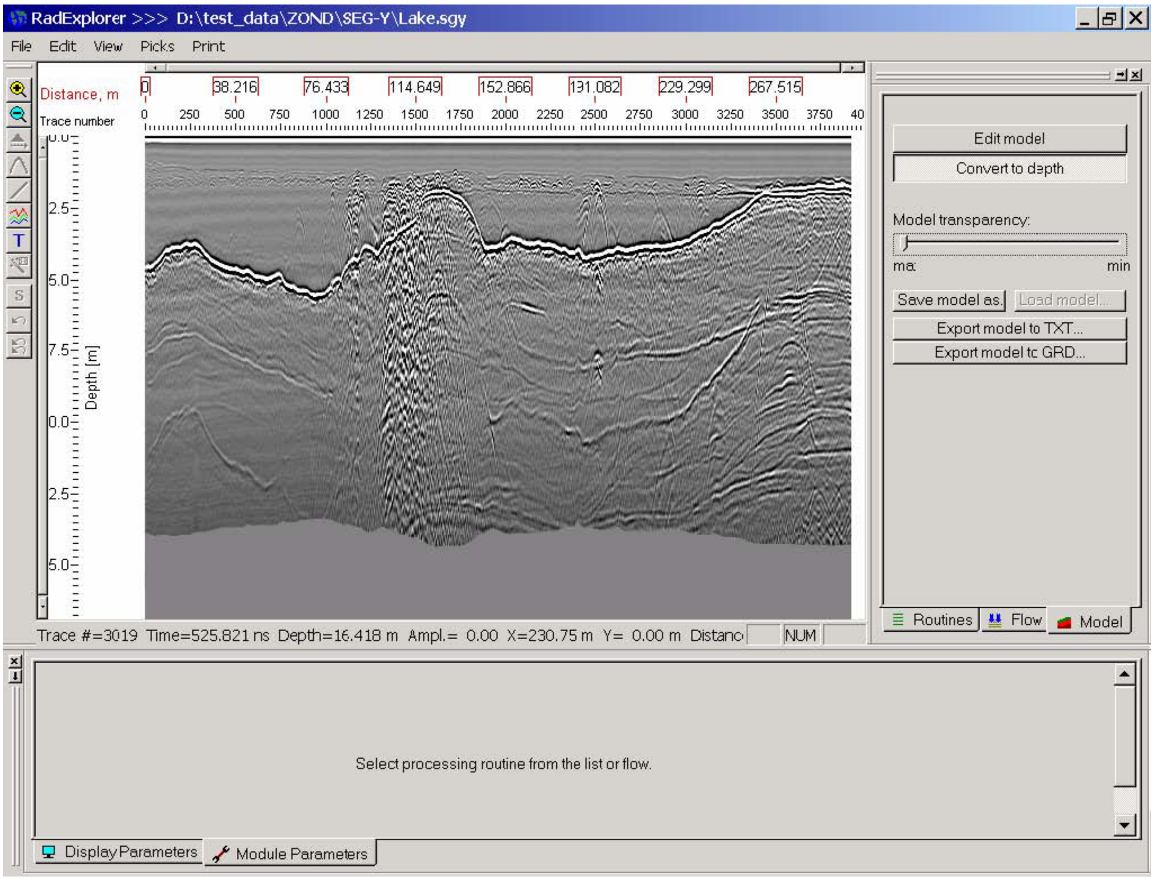


Fig. 5.5.35 – the result of radargram converting to depth in accordance with the model

# Appendix

## Recommended literature

*Hatton, L., Worthington, M.H., Makin, J., 1986. Seismic data processing: theory and practice. Oxford: Blackwell Scientific.*

## System requirements

**Minimum:** Pentium 166, 128 Mb RAM, SVGA, Windows 98/2000/Me/XP

**Recommended minimum:** Pentium 4, 512 Mb RAM, SVGA 1024x768, Windows 2000

## Text export formats of interpretation elements

### Model:

The first line in file describes table content. When model is exported, either dielectric constant or velocity values are saved at indicated distance and depth increments. Beside these values, the file contains X and Y coordinates, current distance (along the profile) and depth. Values in the table are space delimited.

*Example:*

X(m) Y(m) Dist(m) Depth(m) Epsilon
0.0000 0.0000 0.0000 0.0000 9.000000
0.0000 0.0000 0.0000 2.0000 9.000000
0.0000 0.0000 0.0000 4.0000 9.000000
0.0000 0.0000 0.0000 6.0000 9.000000
0.0000 0.0000 0.0000 8.0000 9.000000
3.0000 0.0000 3.0000 0.0000 9.000000
3.0000 0.0000 3.0000 2.0000 1.070155
3.0000 0.0000 3.0000 4.0000 1.070155
3.0000 0.0000 3.0000 6.0000 1.070155
3.0000 0.0000 3.0000 8.0000 9.000000
6.0000 0.0000 6.0000 0.0000 9.000000
6.0000 0.0000 6.0000 2.0000 1.070155
6.0000 0.0000 6.0000 4.0000 1.070155
6.0000 0.0000 6.0000 6.0000 1.070155
6.0000 0.0000 6.0000 8.0000 9.000000
9.0000 0.0000 9.0000 0.0000 9.000000
9.0000 0.0000 9.0000 2.0000 1.070155
9.0000 0.0000 9.0000 4.0000 1.070155
9.0000 0.0000 9.0000 6.0000 1.070155
9.0000 0.0000 9.0000 8.0000 9.000000

## Horizon picks:

When a pick is saved (**Picks/Save as…** menu command), the resulting text file contains the following information:

1. The first line contains the name of the pick.
2. The second line describes the table columns. The column names are separated by colon. In the example below the pick table contains X and Y coordinates, trace number, two-way time, and depth calculated from the time according to current velocity model.
3. Then there is an empty line that is followed by the number of points (nodes) in the pick (in the example below the pick consists of 9 points).
4. The following line contains some service information: line type code, line width, colour code.
5. Then the space-delimited values follow. Each node of the pick correspond to a raw in the table. The order of the values in a raw corresponds to what is described in the second line of the file.

*Example:*

Horizon1
REC_X:REC_Y:TRACENO:TIME:DEPTH

9
0 2 200

2.070000 0.000000 69 20.322809 1.016140
3.540000 0.000000 118 19.091124 0.954556
5.220000 0.000000 174 20.322809 1.016140
6.120000 0.000000 204 22.991461 1.234545
8.160000 0.000000 272 23.812584 1.337821
9.540000 0.000000 318 18.885843 0.944292
10.740000 0.000000 358 18.680561 0.934028
12.179999 0.000000 406 22.991461 1.149573
14.009999 0.000000 467 21.965055 1.098253

Besides, when several picks are available the software allows exporting all of them into single table file (**Picks/Export pick table…** menu command). Here, at indicated distance increment the following values are saved: two-way time or depth, current distance, X and Y coordinates and geographic coordinates when available.

If the two-way time values are saved, the first line of the file contains "TWT", otherwise – "Depth". Then, after an empty line, there is a line describing table columns. This is followed by raws with space-delimited values. If at a certain distance some value does not exist, "N/A" is placed instead of a number. When the pick table is exported the picks appear in the same order as the were created by the software.

*Example:*

TWT

Distance X Y Lat_D Lon_D Horizon1 Horizon2
0.0000 0.0000 0.0000 N/A N/A N/A N/A
3.0000 3.0000 0.0000 N/A N/A 19.5436 32.2291
6.0000 6.0000 0.0000 N/A N/A 22.6356 35.5400
9.0000 9.0000 0.0000 N/A N/A 20.8137 33.5292
12.0000 12.0000 0.0000 N/A N/A 22.4526 N/A
15.0000 15.0000 0.0000 N/A N/A N/A N/A

## Info-marks:

When info-marks are exported, the file contains exactly the strings that has been displayed on the screen. Separate info-marks are divided by asterisks.

*Example:*

```
******************
Pipe1
X=  3.42 m  Y=  0.00 m
Dist=  3.42 m
Depth=1.129 m

******************
Pipe2
Trc#= 330
```

Depth=0.657 m

### *SEG-Y format used by RadExplorer software*

The SEG-Y format is an international standard format designed by the *Society of Exploration Geophysicists (SEG)* primarily for the needs of seismics and acoustics that is supported by most of the data processing systems (both seismic and GPR). The detailed description of SEG-Y format can be viewed on the web-site of the *Society of Exploration Geophysicists (SEG)* in *"Technical Standards"* section: http://seg.org/publications/tech-stand/ .

However, since the format does not fully suit to GPR needs, few minor modifications to the standard have been implemented:

1. All time-based values (i.e. sample interval and delay) are represented in PICOsecons, not in MICROsecons or MILLIseconds.
2. Any trace may have so-called mark – an integer number. This mark is placed into 239-240 bytes of the Trace Header. When a trace has a mark, bytes 237-238 contain a mark indicator equal to 55H.
3. The coordinates (stored in standard SEG-Y fields, bytes 73-88) are always supposed to be measured in meters. (If you wish to store geographical coordinate there for data exchange purposes you may do it through the *SEG-Y Output* **Advanced** dialog. However, be aware of the fact that these coordinates will not be properly recovered by **RadExplorer** when you re-open the file).
4. The geographic coordinates, when available, are saved to bytes 229-232 (longitude) and 233-236 (latitude). The values are saved as decimal degrees multiplied by $10^6$ and within the format are stored as signed long integer numbers (4-byte long). In this case, in bytes 227-228 is recorded 1, otherwise – 0 (this is a flag of geographic coordinates availability).

It is important to be aware of these modifications in case of data exchange with other processing software.