



Refraction Seismic Survey Data Processing in RadExPro Using the Easy Refraction Module – Practical Guide

(Revised April, 5, 2011)

DECO Geophysical SC 000
Moscow State University Science Park,
1-77 Leninskie Gory
Moscow 119992, Russia
Tel./Fax: (+7 495) 930 94 14
E-mail: support@radexpro.ru
Website: www.radexpro.com

Contents

Contents.....	2
Introduction.....	2
Creating a project.....	2
Data loading and geometry assignment.....	6
First arrivals picking.....	12
Working with the Easy Refraction module.....	15
Identifying travel time curve fragments related to different layers.....	17
Automatic travel time inversion.....	19
Manual travel time inversion.....	20
Exporting the results.....	23

Introduction

This Guide is intended for users making their first steps in refraction seismic survey (RSS) data processing in RadExPro using the Easy Refraction module. The Guide covers all processing stages – from data loading and geometry assignment to first arrival picking, identification of travel time curve sections corresponding to different layers, and, finally, travel time inversion and generation of a layered velocity model of the medium.

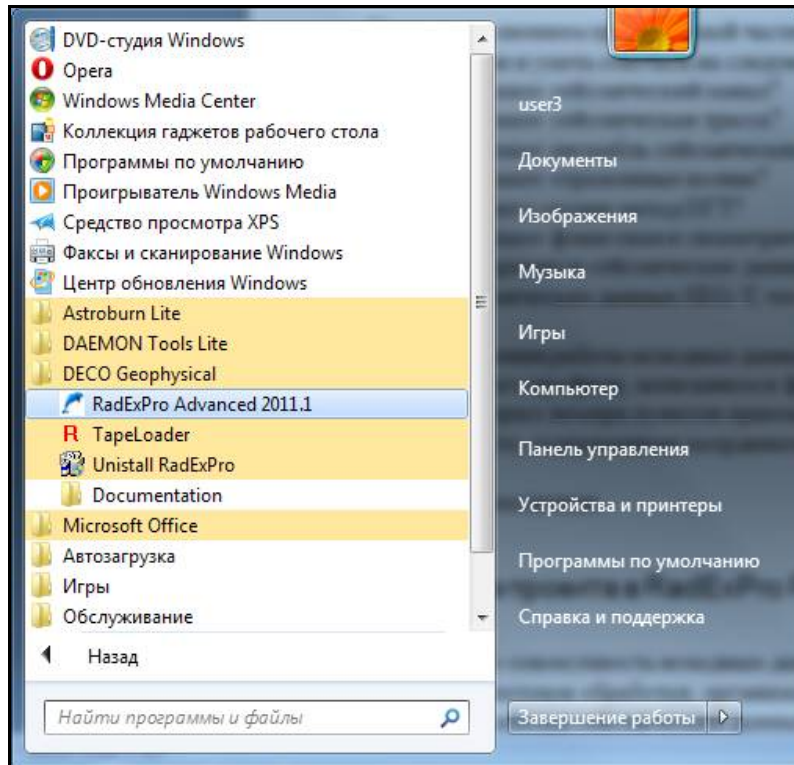
It is assumed that the user is already familiar with the theory behind RSS and the t₀ method.

Source data as well as the project that should be generated as a result of completing this tutorial can be downloaded from our website: <http://www.radexpro.com/downloads/tutorials>

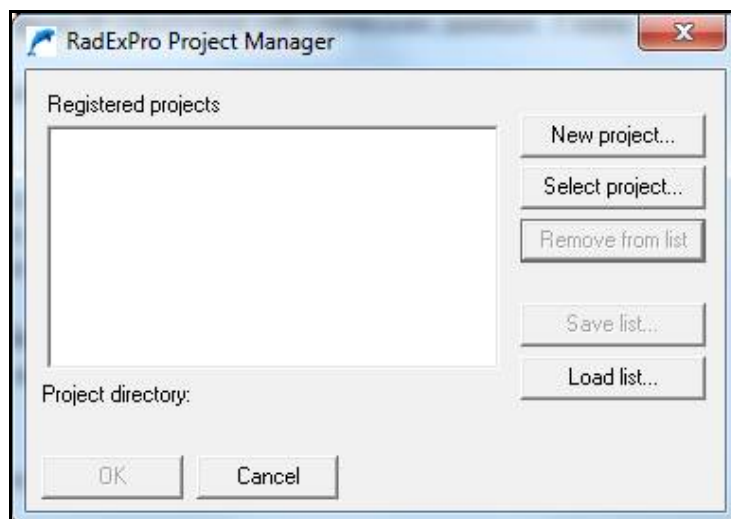
Creating a project

A project is a combination of source data, intermediate and final processing results, and processing flows organized into a common database used by RadExPro seismic data processing package. Projects are stored in separate directories on the hard disk. When a new project is created, a project directory is automatically created for it. Projects can be moved between computers by simply copying the appropriate directory (provided that all used data are stored within that directory).

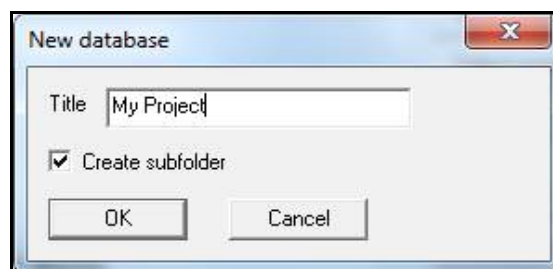
Launch the project manager by opening the Windows Start Menu and selecting All Programs/DECO Geophysical/RadExPro 2011.2.



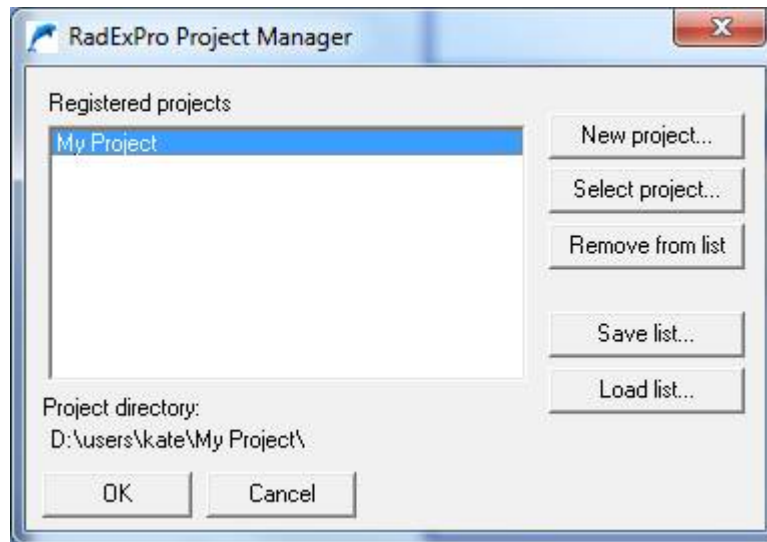
Launching the project manager opens a dialog box with a list of registered projects.



Click the **New Project** button and select a parent directory on the hard disk where the project subdirectory will be created. Another dialog box will appear, prompting you to enter a project name.



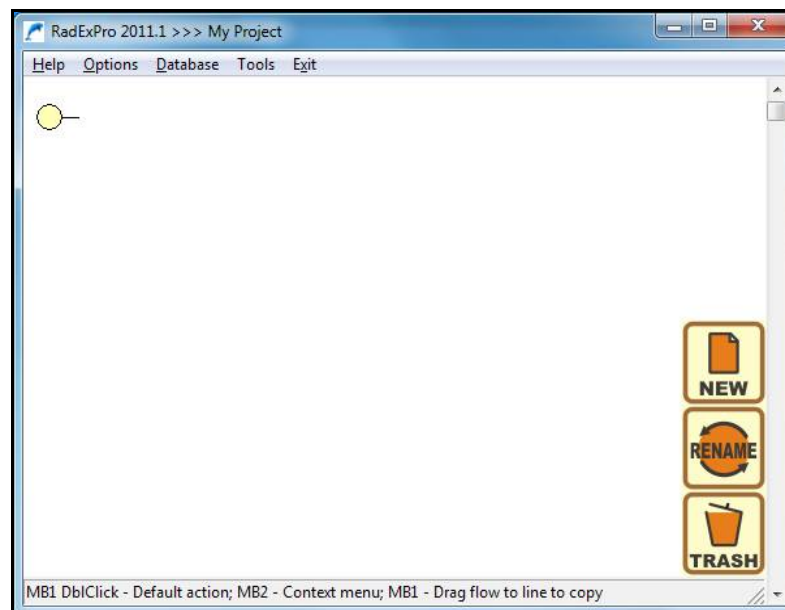
Make sure that the **Create subfolder** option is checked and press **Ok**. A subdirectory with the same name as the project will be created in the selected directory. The project will also appear in the list of available (registered) projects.



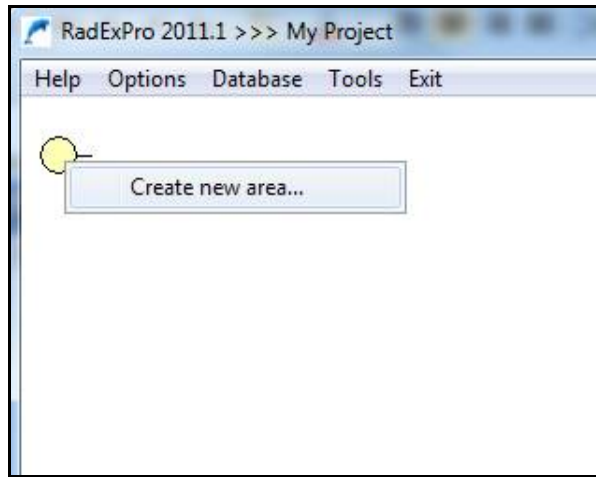
Select the project and press **Ok**.

This will open the main RadExPro window containing the project tree.

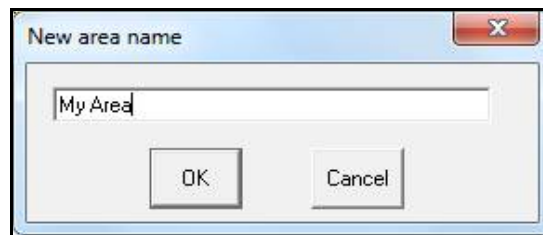
Before starting to work on the project, we recommend creating a directory called **data** within the project directory and copying all data to it. Although this step is optional (data located outside the project can also be read), storing the data within the project directory allows the program to use relative file paths rather than absolute ones. This makes project migration from one computer to another easier.



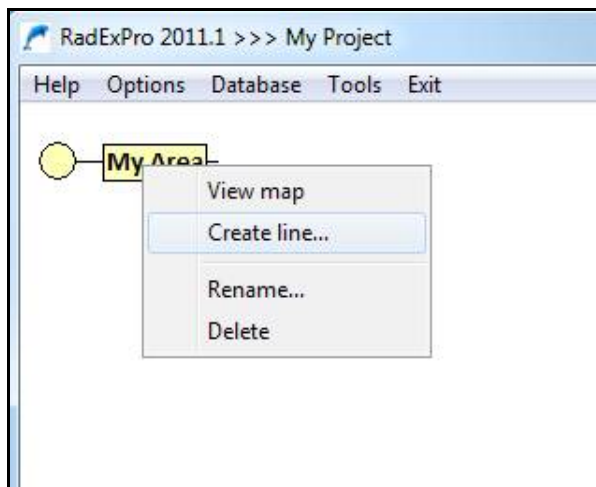
The RadExPro database has 3 structural levels. The upper level corresponds to the project area, the middle level – to the line, and the lower level – to the processing flow. Right-click the yellow circle, select the **Create new area** option, and enter a name for the project area.



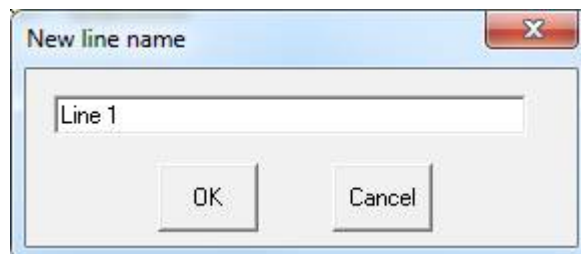
The picture below shows the dialog box prompting you to enter the area name:



In a similar manner, right-click the yellow rectangle with the area name, select **Create line**, and create a new line.



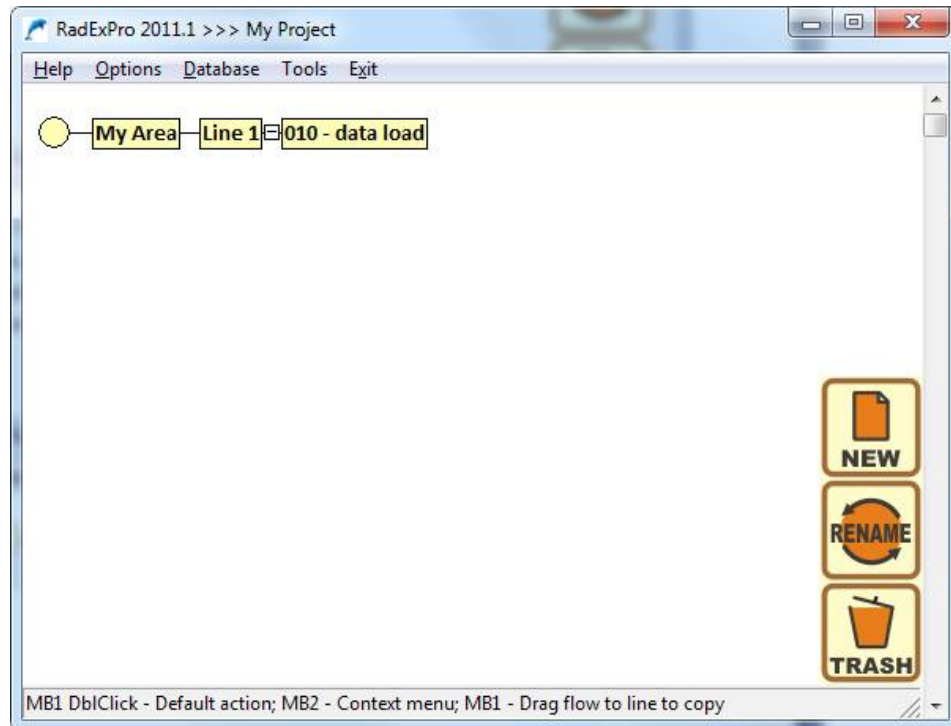
Enter the line name just like the area name.



The database allows storing several areas within one project and several lines within each area. Each line is processed by several flows.

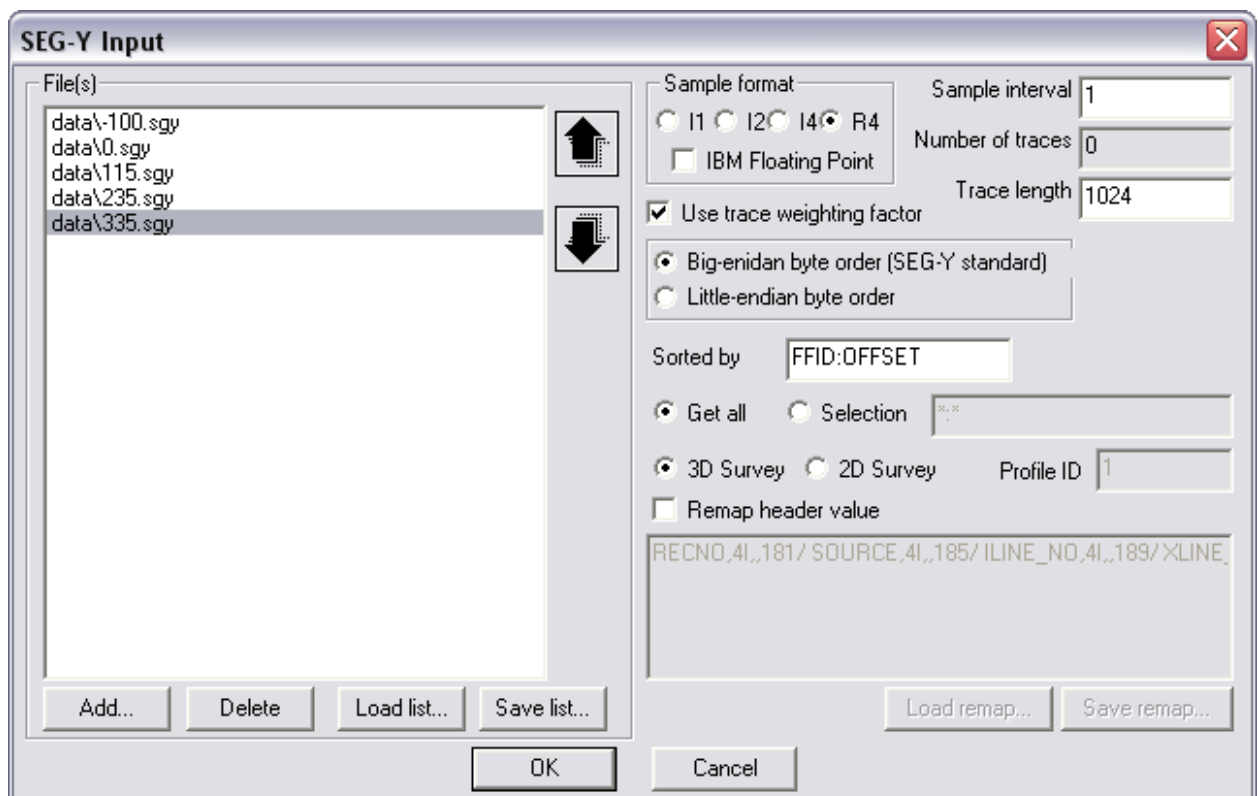
Data loading and geometry assignment

Create a processing flow named *010 – data load* in the same manner as you created the area and the line.

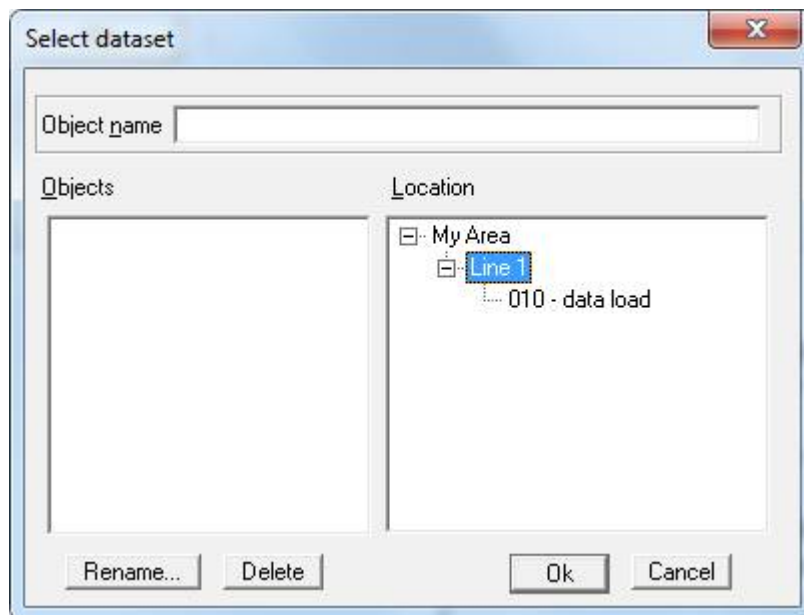


Switch to the flow editing mode by double-clicking the flow name with the left mouse button. This will open the flow editor window where we will now create a flow consisting of the SEG-Y Input and Trace Output modules.

Specify the data reading parameters when adding the SEG-Y Input module.

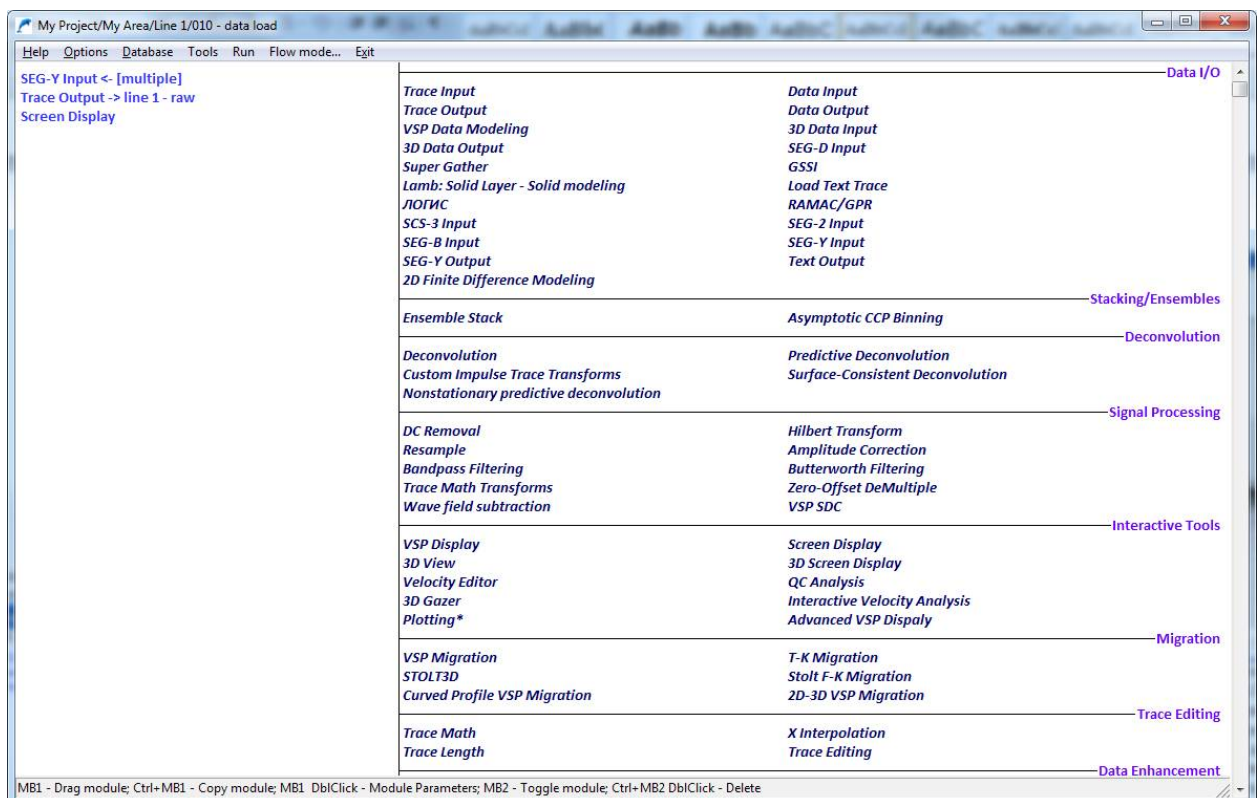


After adding the SEG-Y Input module add the Trace Output module to the flow. This module will save the read data to the database so that they can have geometry assigned to them at a later time. Name the object that will contain these data *line 1 – raw* and place it at the second database level in *Line 1* (as shown in the picture below).

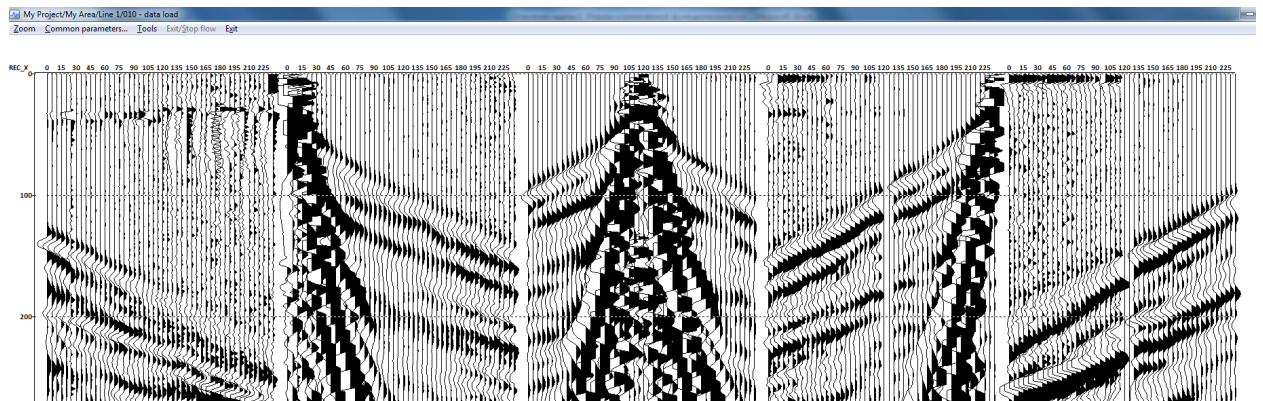


Also, add the Screen Display module to the flow after the Trace Output module for monitoring purposes.

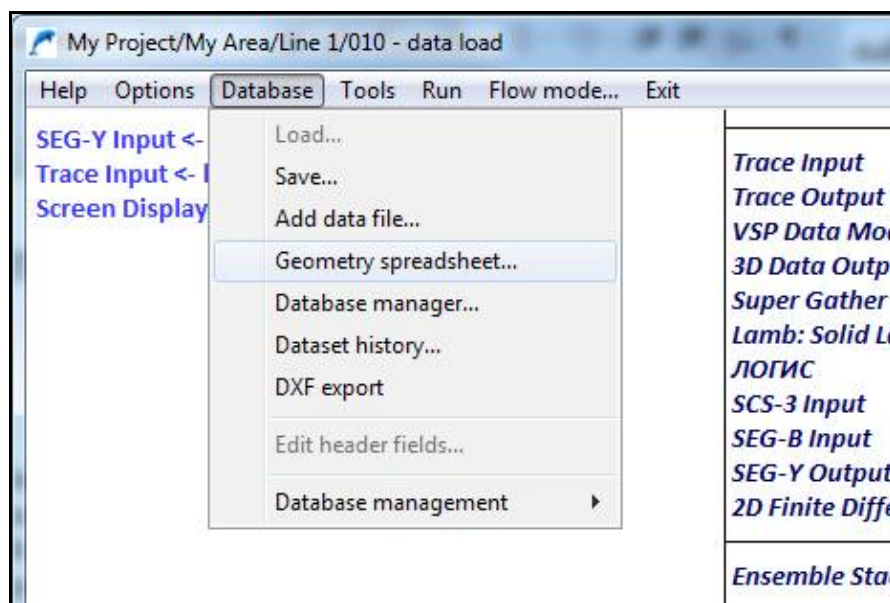
The resulting flow should look like this:



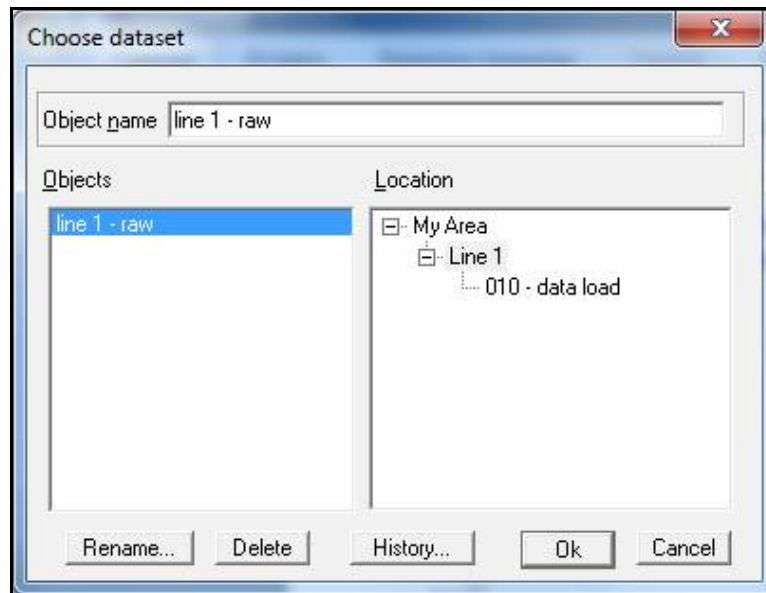
Select the **Run** menu item to run the flow. The Screen Display window showing the data being entered will open, and the data themselves will be read from the file on the hard disk and saved to the database. The Screen Display window that should appear on the screen is shown below.



Now we need to assign the geometry – the source coordinates (SOU_X) and the receiver coordinates (REC_X) – to the seismic data. RadExPro package uses the Geometry Spreadsheet tool to handle seismic data header fields. To open it, select the **Database/Geometry Spreadsheet...** menu item.



Then select the set of seismic data whose geometry you need to edit.



Add header fields SOU_X and REC_X using the **Add field** command in the **Fields** menu. The appearance of the Geometry Spreadsheet window is shown in the picture below.

TRACENO	SOU_X	REC_X
1	-100.00000	0.00000
2	-100.00000	5.00000
3	-100.00000	10.00000
4	-100.00000	15.00000
5	-100.00000	20.00000
6	-100.00000	25.00000
7	-100.00000	30.00000
8	-100.00000	35.00000
9	-100.00000	40.00000
10	-100.00000	45.00000
11	-100.00000	50.00000
12	-100.00000	55.00000
13	-100.00000	60.00000
14	-100.00000	65.00000
15	-100.00000	70.00000
16	-100.00000	75.00000
17	-100.00000	80.00000
18	-100.00000	85.00000
19	-100.00000	90.00000
20	-100.00000	95.00000
21	-100.00000	100.00000
22	-100.00000	105.00000
23	-100.00000	110.00000
24	-100.00000	115.00000

For the sake of simplicity, in this training example the geometry has already been written in the source files and read from them by the SEG-Y Input module. However, if this is not the case in real life, geometry can be assigned in the Geometry Spreadsheet window using the Trace Header Math mathematical operation editor available in the **Tools/Header Math** menu of the **Geometry Spreadsheet** window.

Let us analyze the geometry input procedure for the standard 48-channel RSS array with a 5 m interval between the receivers and several source positions placed at irregular intervals (48 channels of input data).

In this example we assume that only the trace number field – Traceno – is initially written in the headers. In our case there are 48 traces per SP. Based on this, let us calculate the SP number (FFID header) as follows:

$$ffid = trunc([(traceno]-1)/48]+1$$

where the trunc (x) function returns the value of the nearest integer that is less or equal to x. As a result, each set of 48 traces will have a corresponding individual ffid number.

Now let us number the channels – using the traceno field as well:

$$chan = fmod([traceno]-1,48)+1$$

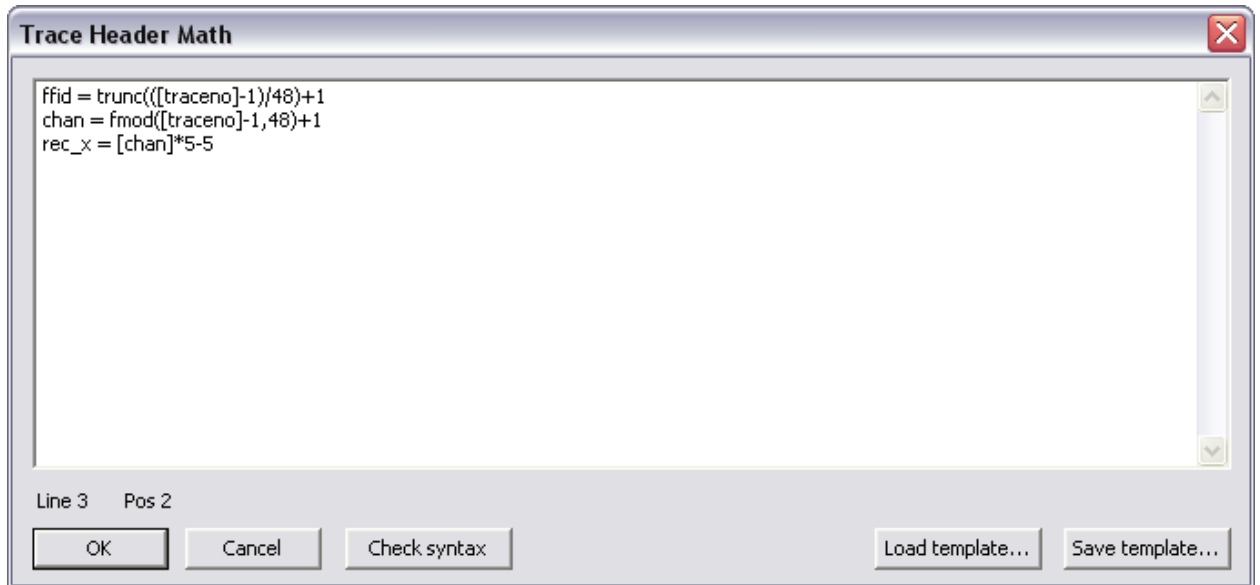
where the $fmod(x,y)$ function returns the remainder of division of x by y . As a result, channels will be numbered 1 through 48 for each $ffid$ set.

The next step is to fill in the sou_x and rec_x coordinates using the array data.

Receiver coordinates can be filled in relatively easily – all you need to do is multiply the channel number by the interval, i.e.:

$$rec_x = [chan]*5-5$$

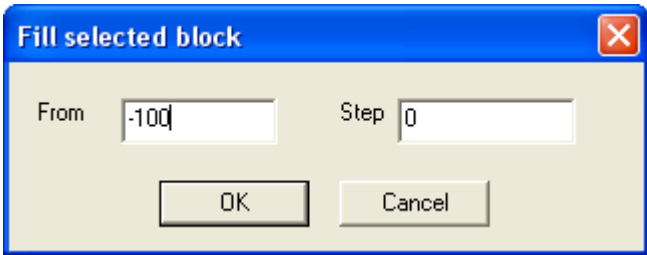
The resulting Header Math window used to assign the geometry should look like this:



If the first and the last SP are outside the array (offset), and there are different intervals between SP, it is easier to specify the SOU_X source coordinates manually. Let us consider the first SP (FFID 1) example with a -100 offset. Select the sou_x column part corresponding to this SP with the left mouse button:

TRACENO	REC_X	SOU_X	FFID	CHAN
11	50.00000	-100.00000	1	11
12	55.00000	-100.00000	1	12
13	60.00000	-100.00000	1	13
14	65.00000	-100.00000	1	14
15	70.00000	-100.00000	1	15
16	75.00000	-100.00000	1	16
17	80.00000	-100.00000	1	17
18	85.00000	-100.00000	1	18
19	90.00000	-100.00000	1	19
20	95.00000	-100.00000	1	20
21	100.00000	-100.00000	1	21
22	105.00000	-100.00000	1	22
23	110.00000	-100.00000	1	23
24	115.00000	-100.00000	1	24
25	120.00000	-100.00000	1	25
26	125.00000	-100.00000	1	26
27	130.00000	-100.00000	1	27
28	135.00000	-100.00000	1	28
29	140.00000	-100.00000	1	29
30	145.00000	-100.00000	1	30
31	150.00000	-100.00000	1	31
32	155.00000	-100.00000	1	32
33	160.00000	-100.00000	1	33
34	165.00000	-100.00000	1	34
35	170.00000	-100.00000	1	35
36	175.00000	-100.00000	1	36
37	180.00000	-100.00000	1	37
38	185.00000	-100.00000	1	38
39	190.00000	-100.00000	1	39
40	195.00000	-100.00000	1	40
41	200.00000	-100.00000	1	41
42	205.00000	-100.00000	1	42
43	210.00000	-100.00000	1	43
44	215.00000	-100.00000	1	44
45	220.00000	-100.00000	1	45
46	225.00000	-100.00000	1	46
47	230.00000	-100.00000	1	47
48	235.00000	-100.00000	1	48
49	0.00000	0.00000	2	1
50	5.00000	0.00000	2	2

Then press **INS** on your keyboard to open a dialog box. Enter -100 as the initial value and 0 as the step.



The dialog box titled "Fill selected block" has a blue header with a close button. It contains two input fields: "From" with the value "-100" and "Step" with the value "0". Below the fields are "OK" and "Cancel" buttons.

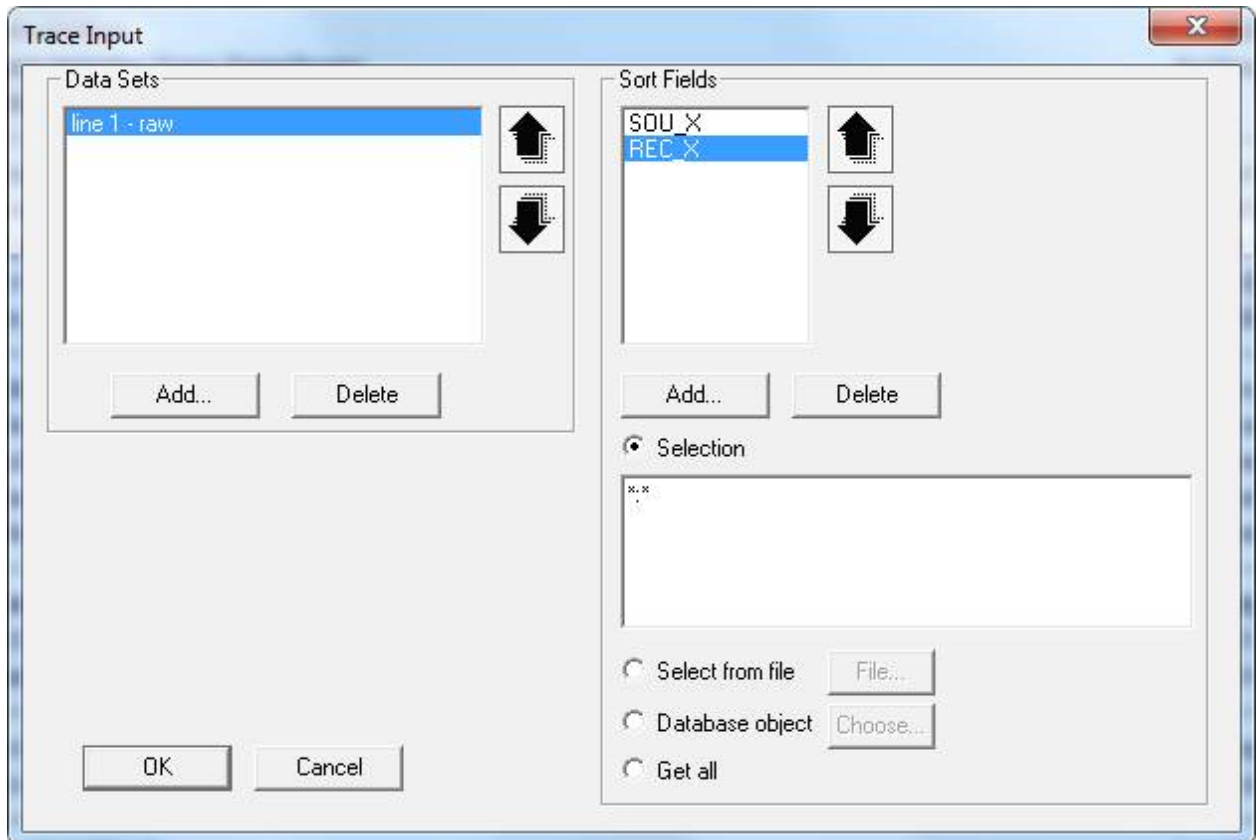
As a result, all selected lines will be filled with -100 value.

Coordinates of other SP can be specified in a similar manner: FFID 2 — SOU_X 0; FFID 3 — SOU_X 115; FFID 4 — SOU_X 235; FFID 5 — SOU_X 335.

You can easily assign geometry to other array types by following the above guidelines and modifying formula parameters as necessary.

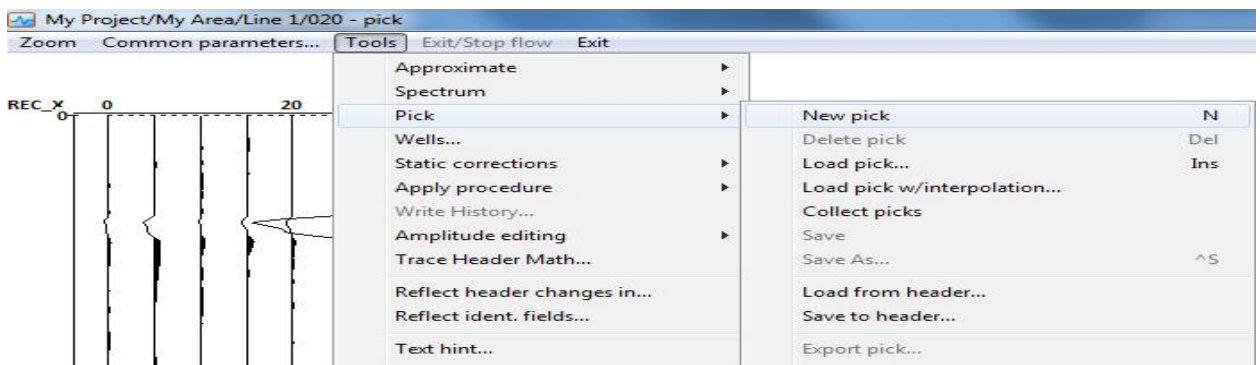
First arrival picking

Create a new flow named *002 – pick* and add the Trace Input module to it. After that add the newly created dataset to the Data Sets window and apply sorting by *SOU_X* and *REC_X* to the entire selection.



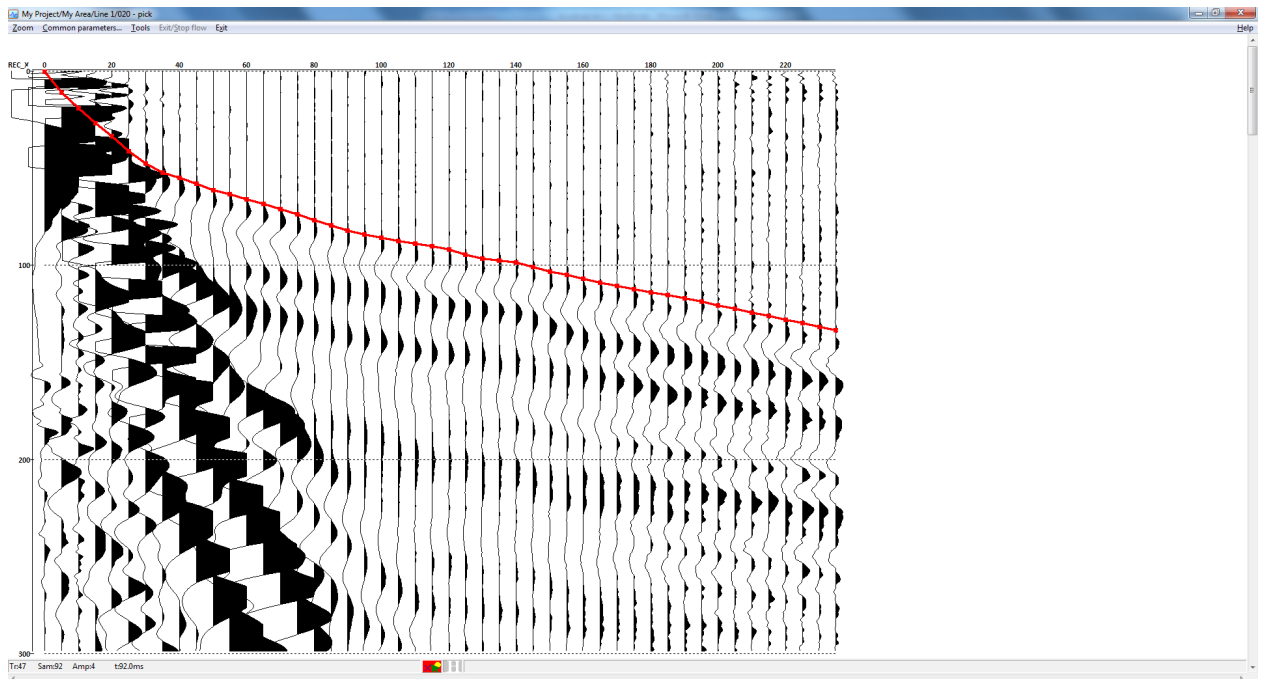
Then add the Screen Display module, selecting trace image scales and amplification factors suitable for first arrivals picking. Additional processing procedures – such as Bandpass Filtering, Hand Static etc. – can be enabled as necessary. However, you should keep in mind that filtering (especially zero-phase) “blurs” first wave arrivals; therefore, first arrivals picking should be done before filtering.

Run the flow. Select the **Tools/New pick** menu item and perform first arrival picking.

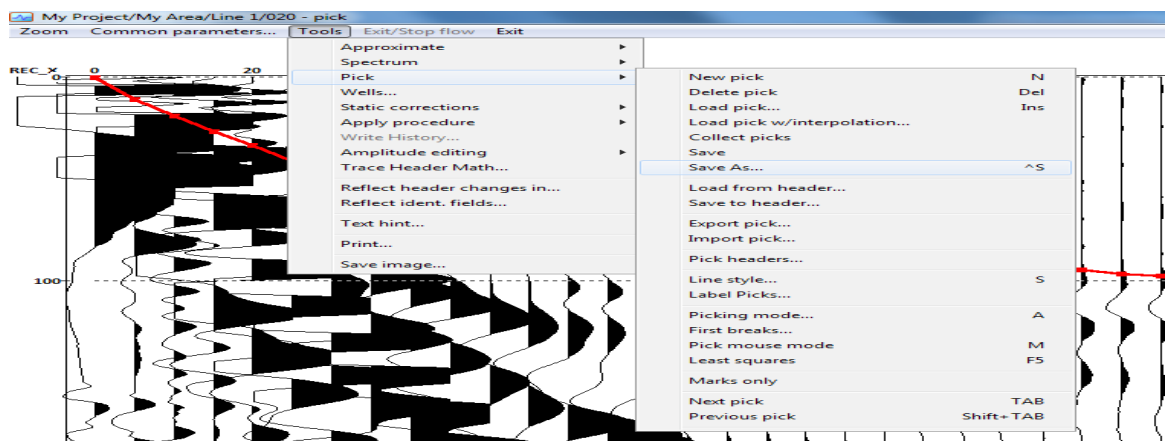


After that you can pick wave arrivals or extremums. Picking can be done manually (**Hand pick**) or semi-automatically (**Auto fill**). In the latter case the program automatically tracks waves between two interpreter picks according to the specified attribute. The picking mode is selected in the pop-up window opened by clicking the **Tools/Pick/Picking mode** menu item (see Help for detailed information). To perform picking, press the left mouse button when the marker is over the selected point. An X mark will appear at that point. Click the left mouse button once more within the same trace to move the X to a

new position, or click within another trace to place a new X. An erroneously placed X can be removed by double-clicking the right mouse button or clicking the right mouse button when the marker is over the X and moving it along the same trace to a point with a different time. If you are not happy with a travel time curve and want to delete it completely, press the **Delete** key.

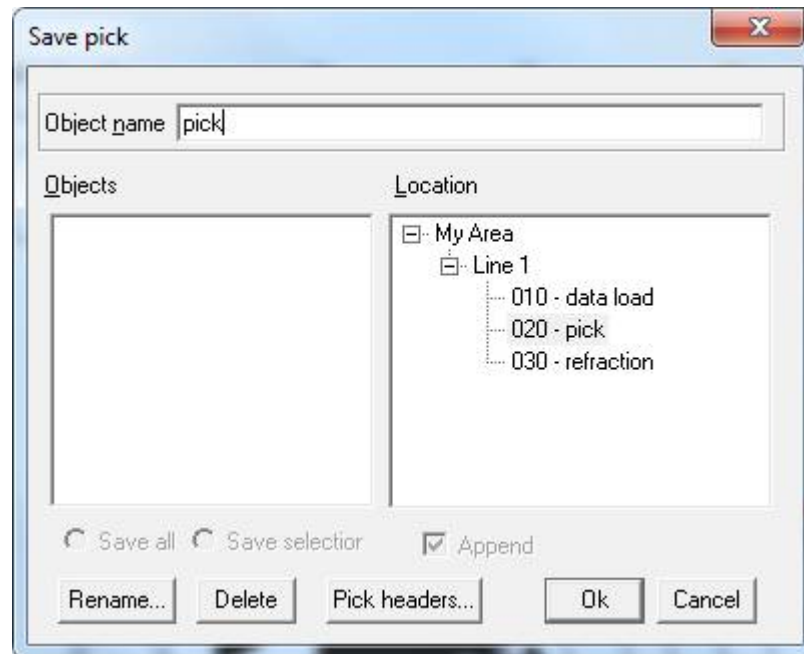


You need to pick all seismograms. To save the pick, select the **Tools/Save As...** menu item.

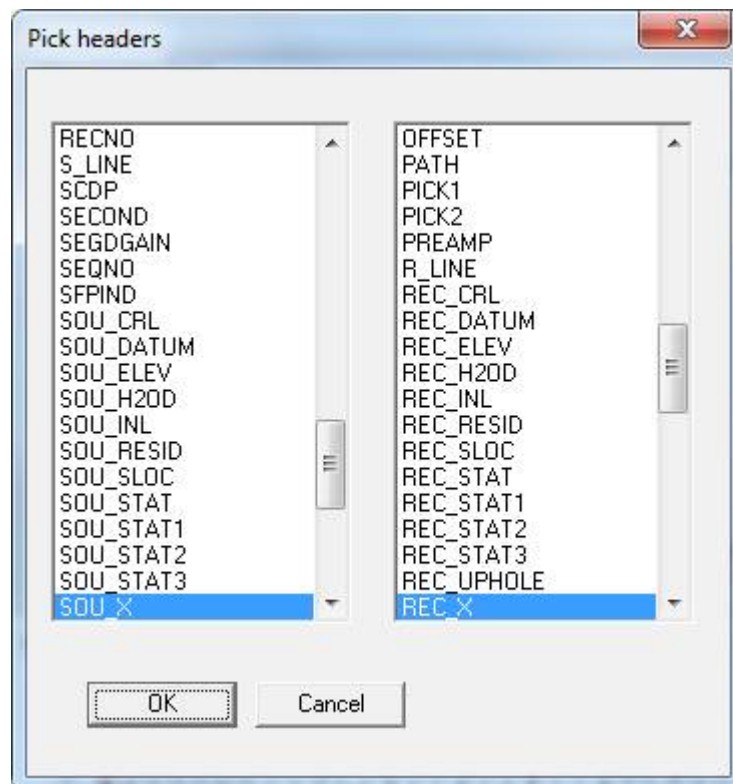


This will open a dialog box where you will be asked to enter the pick name and specify which database object the pick will correspond to by left-clicking the appropriate object.

The program also allows saving travel time curves as text files for further use in other interpretation software (**Tools/Pick/Export pick**).

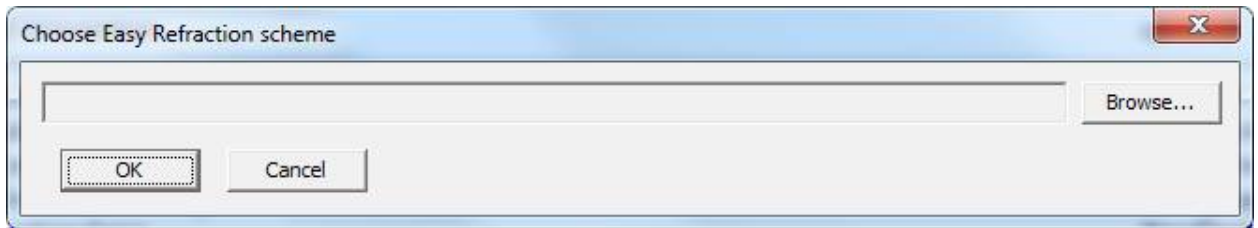


Press **Pick headers...** and make sure that **SOU_X** is selected in the left column and **REC_X** – in the right column.

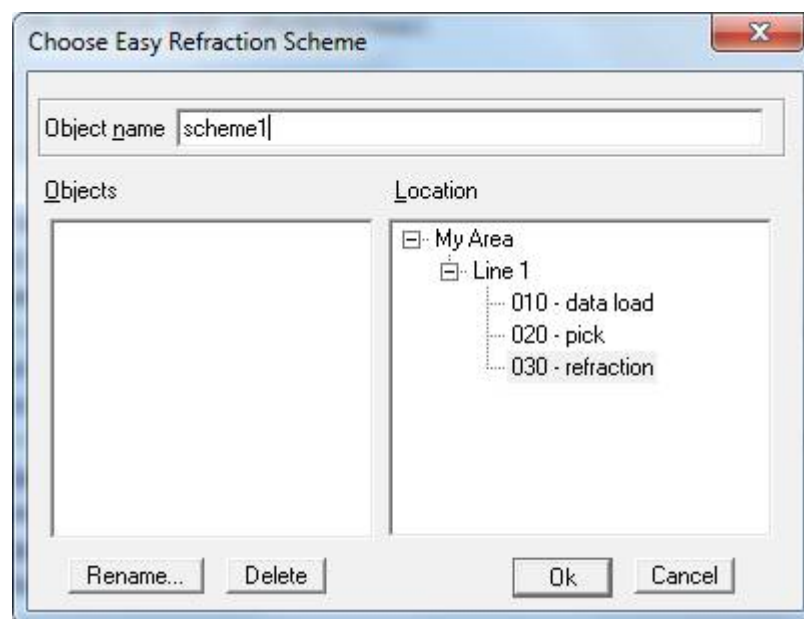


Working with the Easy Refraction module

Create a new flow named *003 – refraction* and add the Easy Refraction module to it. Select **Browse...** in the dialog box.

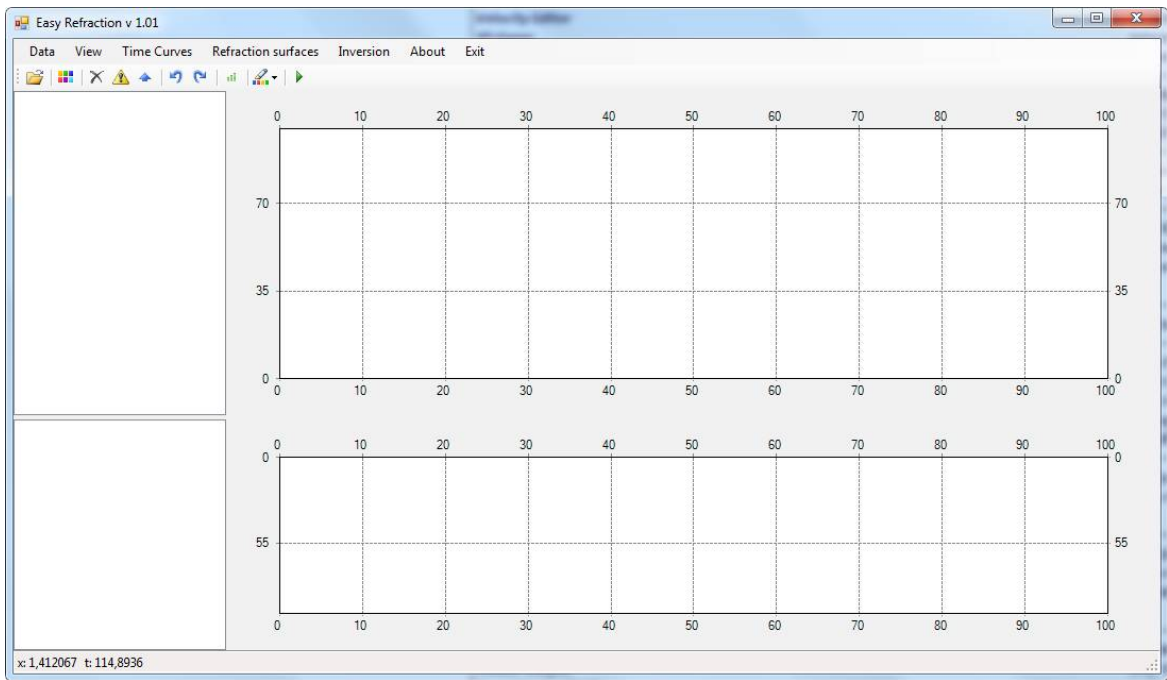


This will open another dialog box prompting us to specify the “scheme” name. An “Easy Refraction scheme” is a combination of travel time curves (possibly divided into segments) corresponding to different layers generated as a result of boundary processing etc. When the user exits the module, its current state is stored in the “scheme”.

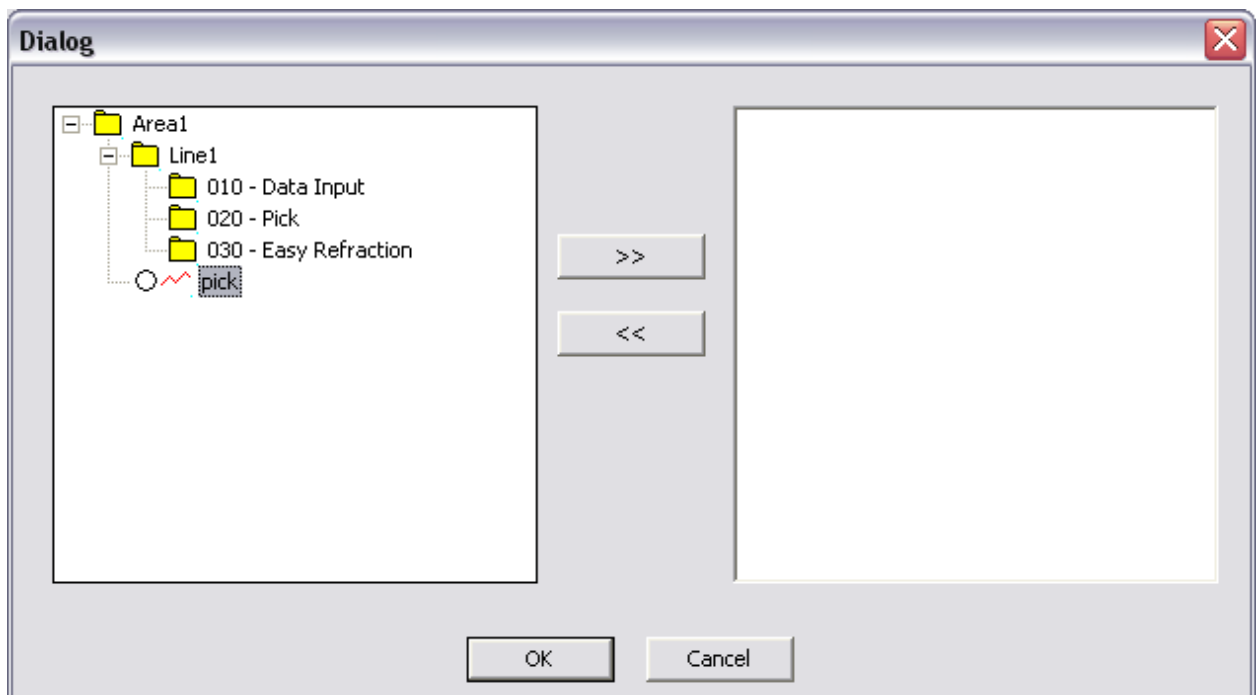


After entering the new scheme name, press **Ok** and run the flow.

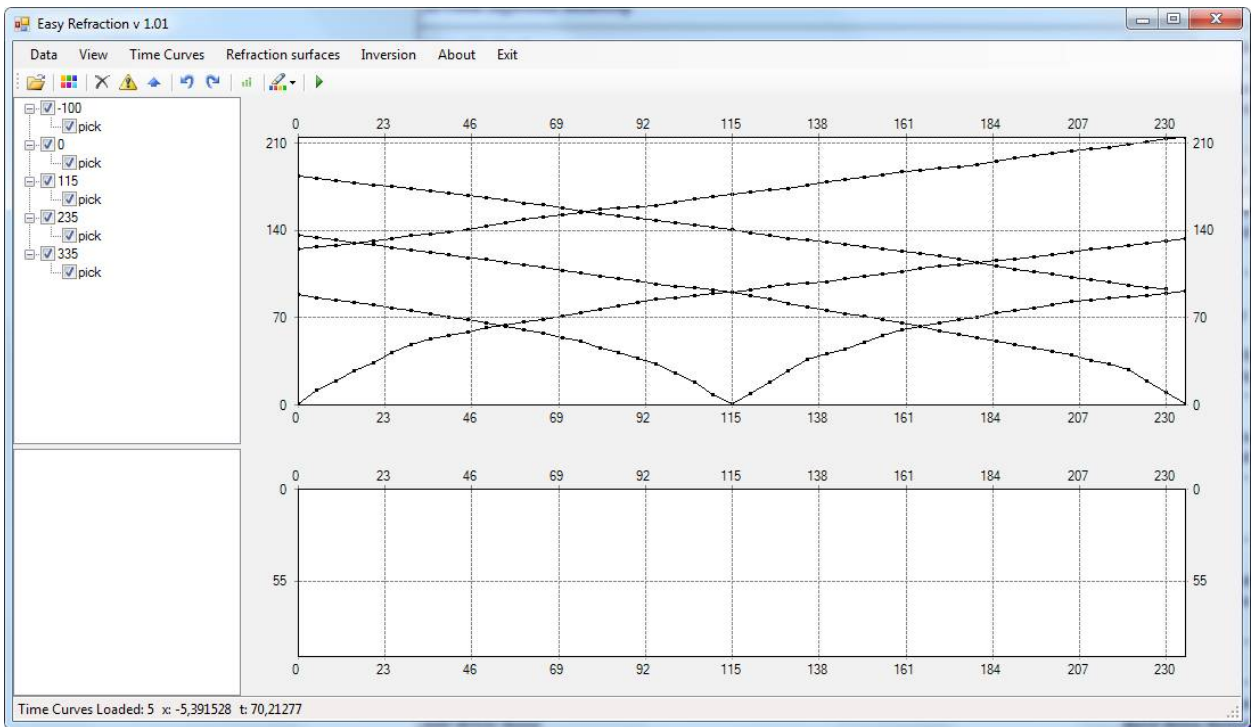
The Easy Refraction module working window will open.



Press the **Load time curves** button to load travel time curves. This will open the travel time curve selection window. Load the necessary time curves and press **Ok**.



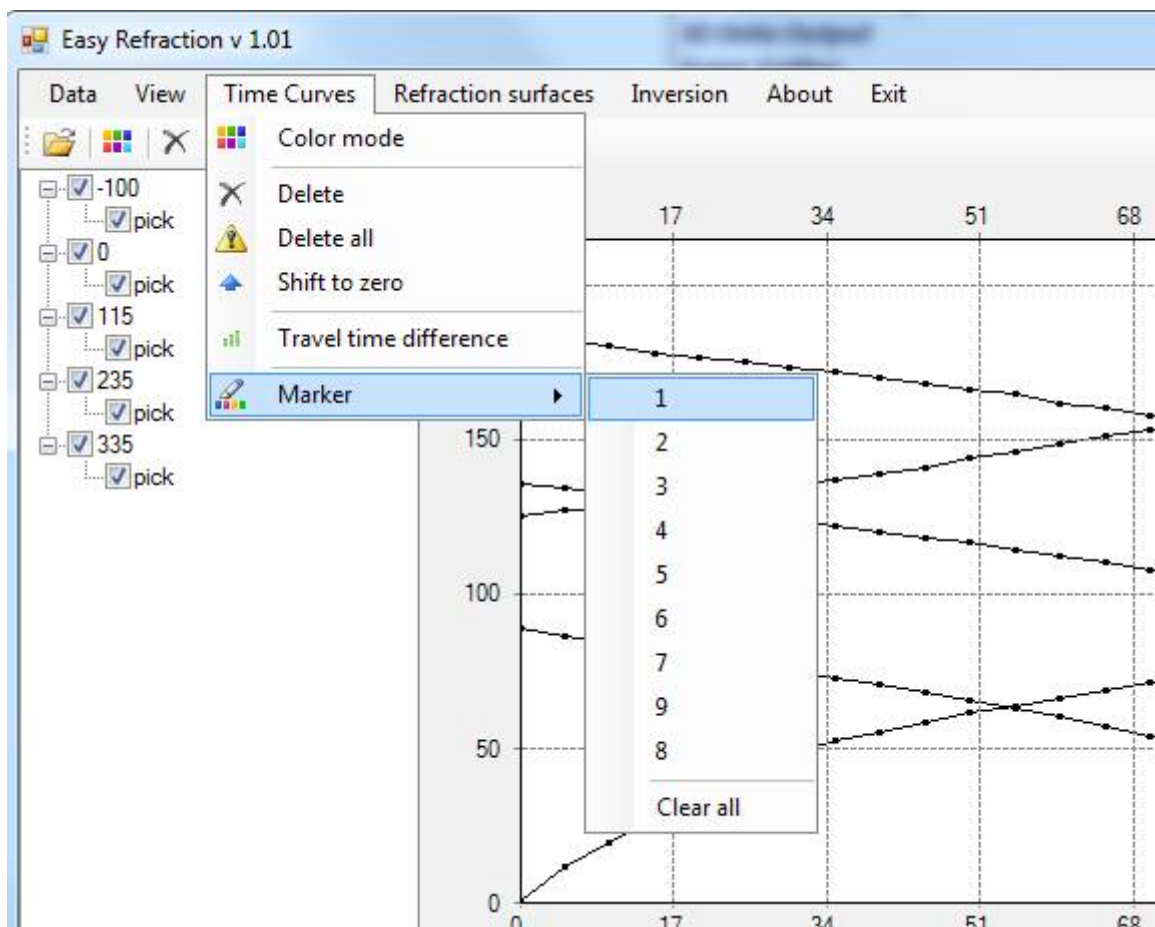
The module window containing the loaded travel time curves will appear.



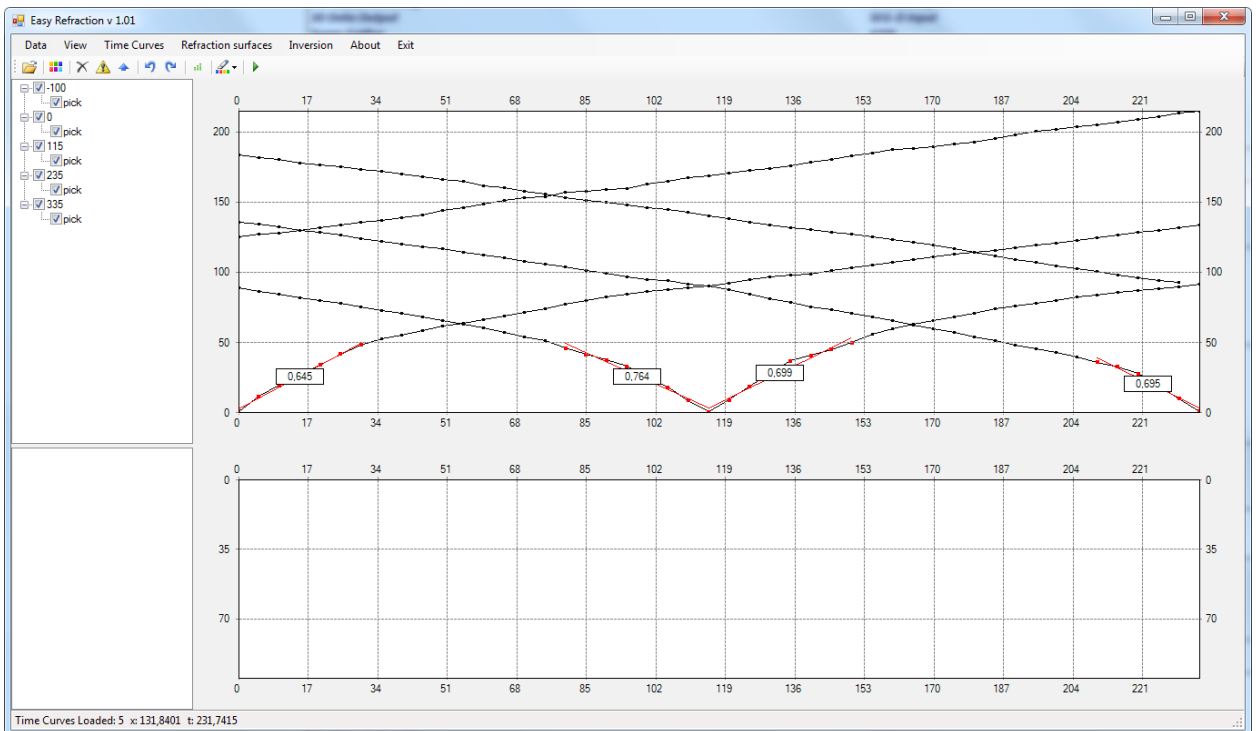
Identifying travel time curve fragments related to different layers

We need to mark travel time curve fragments related to particular layers. Let us do this using the interactive marker.

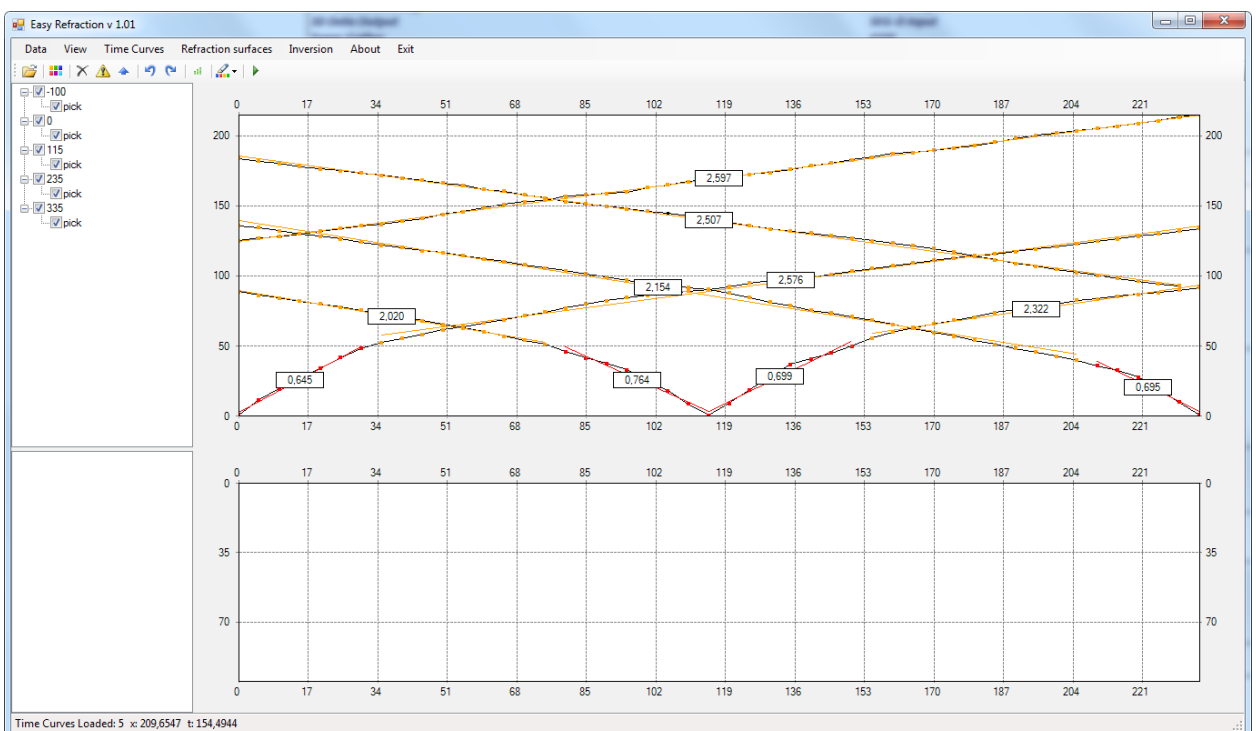
Select the **Time Curves/Marker/1** menu item.



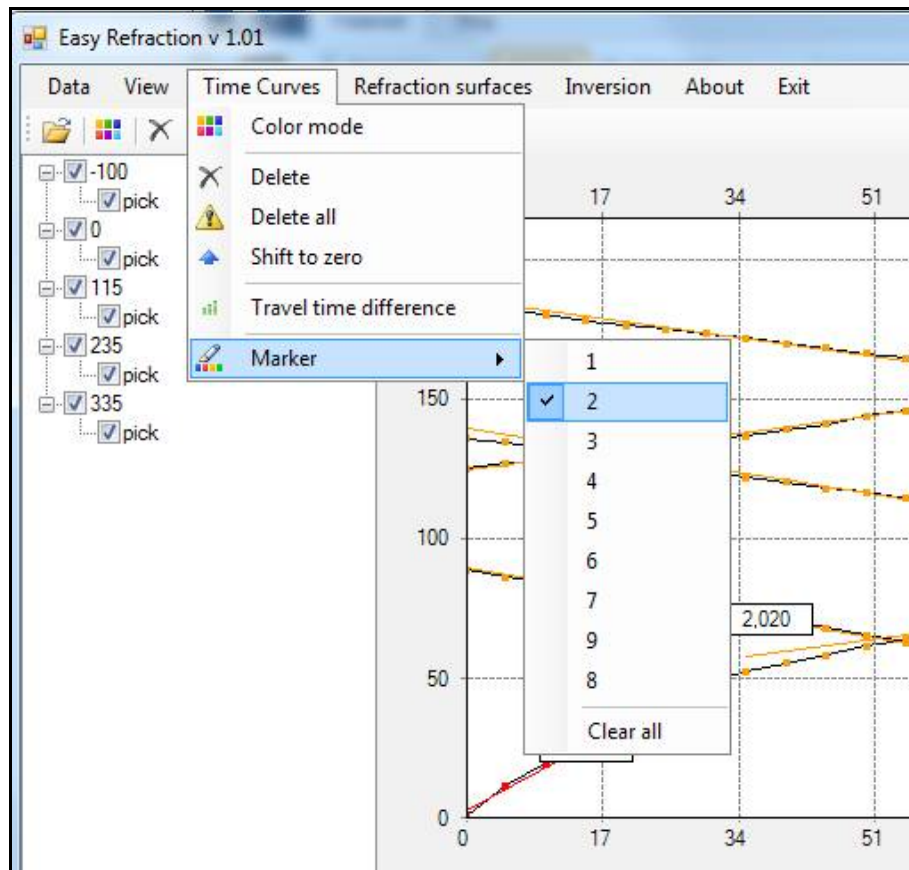
Press and hold the left mouse button to mark the travel time curve sections related to the first layer. The interpreter selects travel time curve breakpoints and determines the number of layers in the section interpretation model according to the principles described in the literature. Linear approximation as well as output of resulting velocity values per travel time curve is done automatically.



Similarly, select the second marker and mark the travel time curve sections related to the second layer. Visualization of individual travel time curves can be disabled by unchecking the appropriate boxes in the left pane of the module.

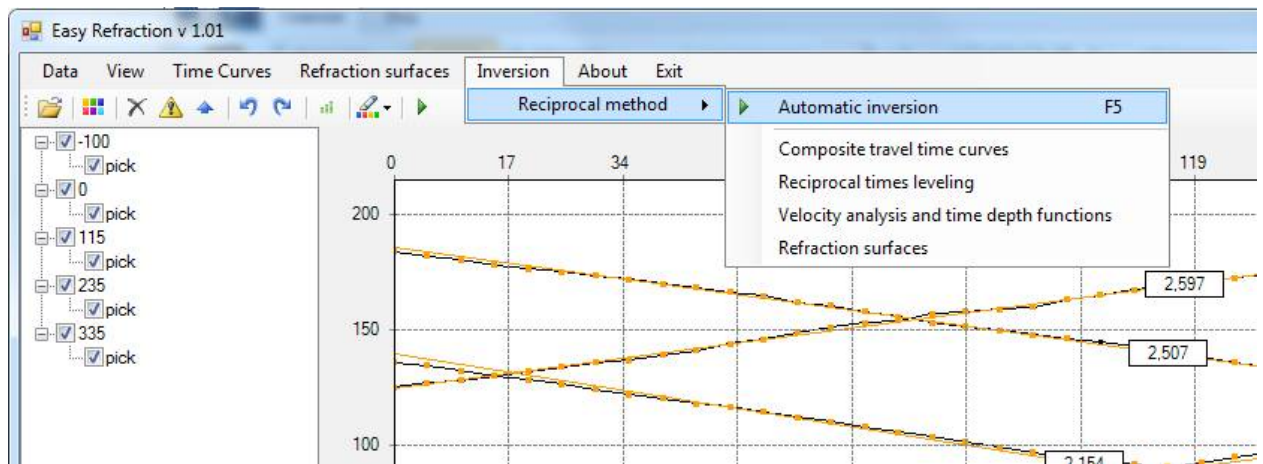


To exit the marker mode, open the **Time Curves/Marker/2** menu item and left-click (or press the **Esc** key on your keyboard) to uncheck it.

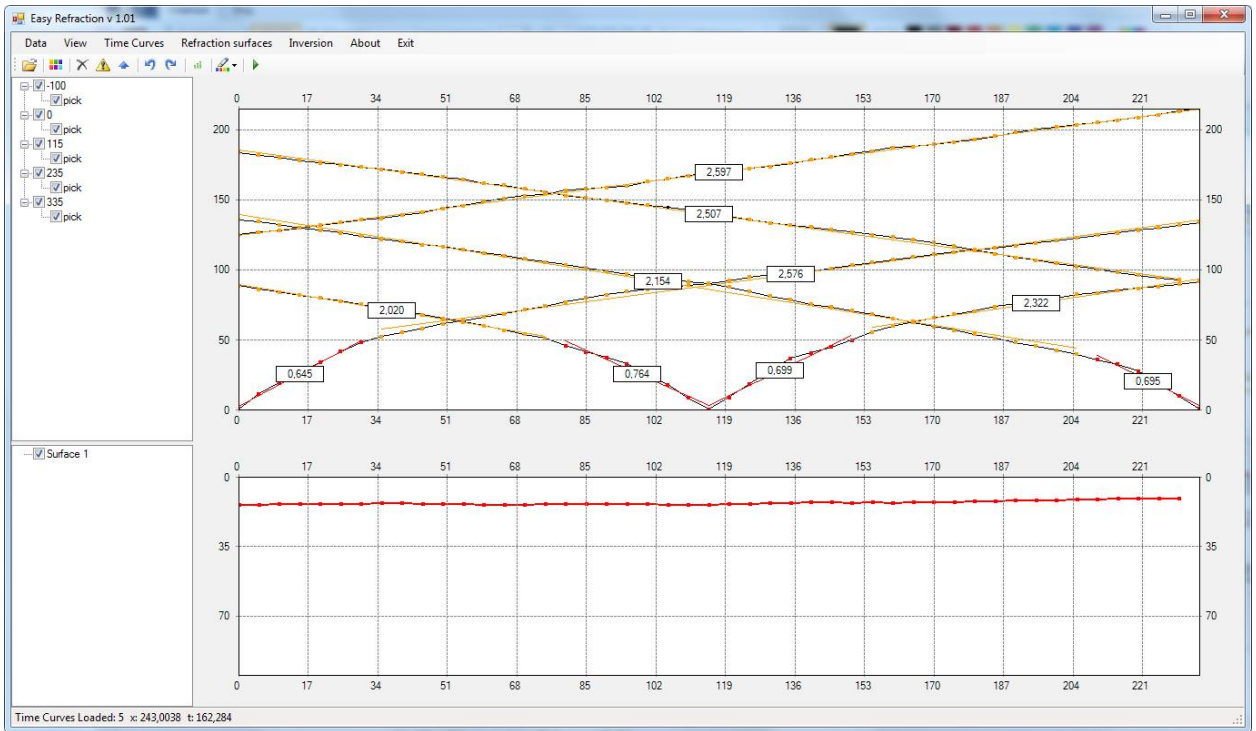


Automatic travel time inversion

Open the **Inversion/Reciprocal Method** menu and select **Automatic inversion**.



The results are shown in the picture below: the position of the boundary between the first and the second layer has been built in the lower part of the module working window.

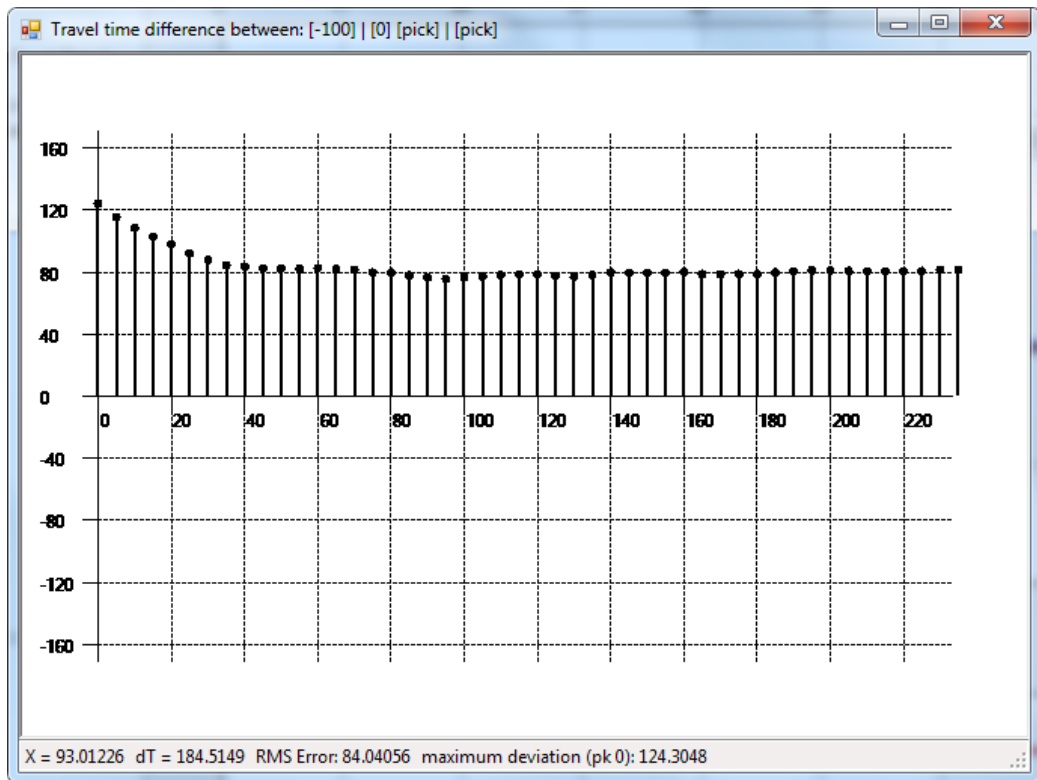


Press **t** to show the layer velocity values at the section.

Manual travel time inversion

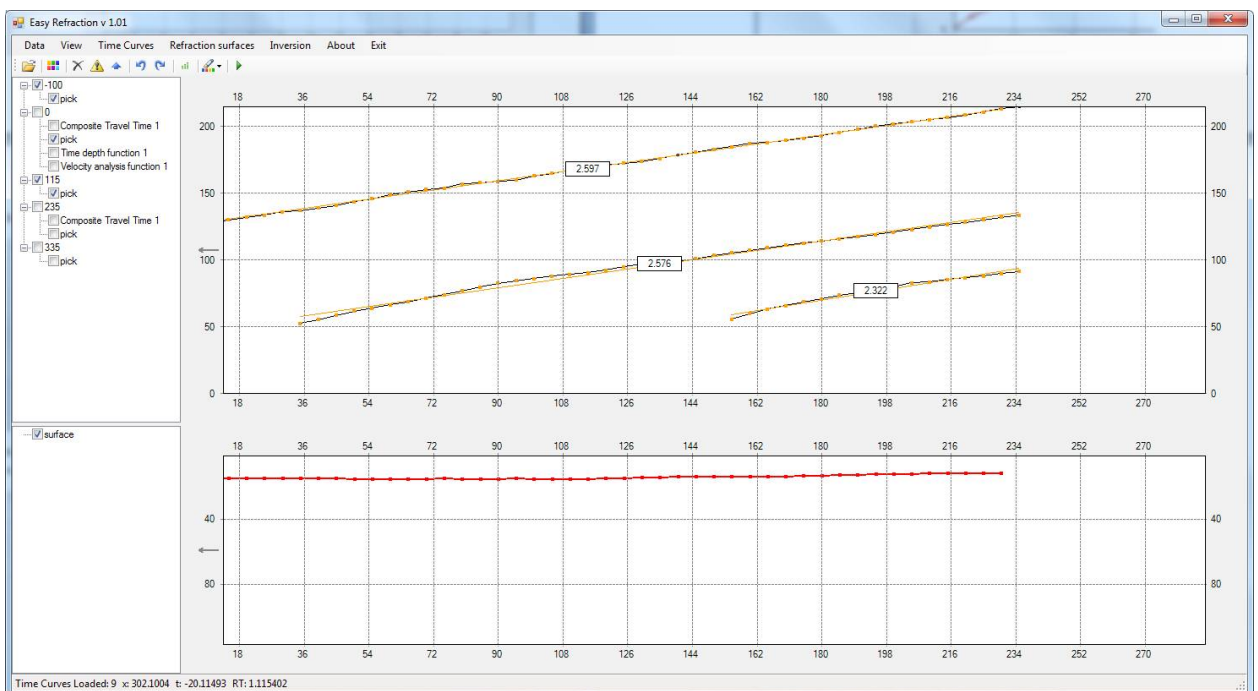
If necessary, all RSS processing stages can be carried out manually.

The module allows building the difference between two travel time curves. To do this, select one travel time curve with the left mouse button, another one – with the right mouse button, and open the **Time Curves/Travel time difference** menu. A graph showing the difference between the two selected travel time curves will be built in a separate window. If diving waves are registered in the first arrival, the entire difference graph will be a decay function; if head waves are registered, the function will decay at first, but then will become constant.



The module allows building composite travel time curves – time curves of waves from each refraction boundary. This procedure is necessary to obtain a travel time curve covering the “dead zone” – the direct wave tracking area on the direct and opposite travel time curves. The head wave travel time curve can be extended into the “dead zone” using catching-up travel time curves – direct and opposite. The procedure is accessed through the **Inversion/Reciprocal Method/Composite travel time curves** menu item.

When building a composite travel time curve, the program takes into account the times of all travel time curves related to the second layer.

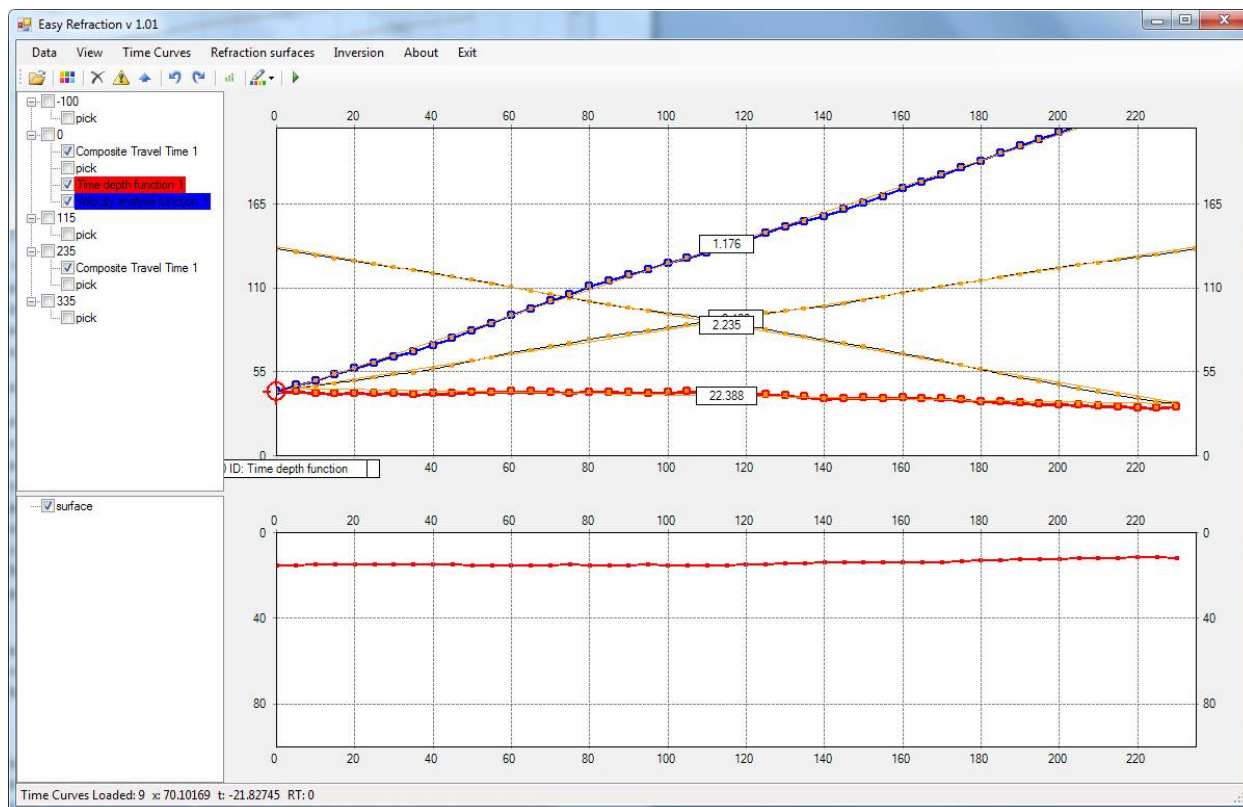


For example, when a composite direct travel time curve is built, only the catching-up travel time curve will be taken into account in the left part, then the average time between the catching-up and direct travel time curve will be accounted for, and finally the average time between all three travel time curves will be factored in. All used travel time curves will be raised or lowered by the time corresponding to the travel time curve located within the array closest to its beginning.

A composite opposite travel time curve is built in a similar manner.

According to the reciprocity principle, the time of travel from the source to the receiver does not change if you swap the source with the receiver. This time corresponds to the reciprocal points on the direct and opposite travel time curves. Therefore, travel time curves need to be tied at the reciprocal points. The module allows viewing the mistie and leveling the reciprocal times. Points to be tied are selected automatically when the user selects two travel time curves using the left and the right mouse button. Reciprocal time (RT) mistie between two reciprocal travel time curves is shown in the lower part of the module window. To tie the reciprocal times, open the **Inversion/Reciprocal Method** menu and select **Reciprocal time leveling**. This function allows finding the average of the reciprocal times and automatically adapts the travel time curves to that average time.

The module also allows building a t_0 travel time curve and a residual travel time curve. To do this, select the direct travel time curve with the left mouse button, the opposite travel time curve – with the right mouse button, and open the **Inversion/Reciprocal Method/Velocity analysis and time depth functions** menu.

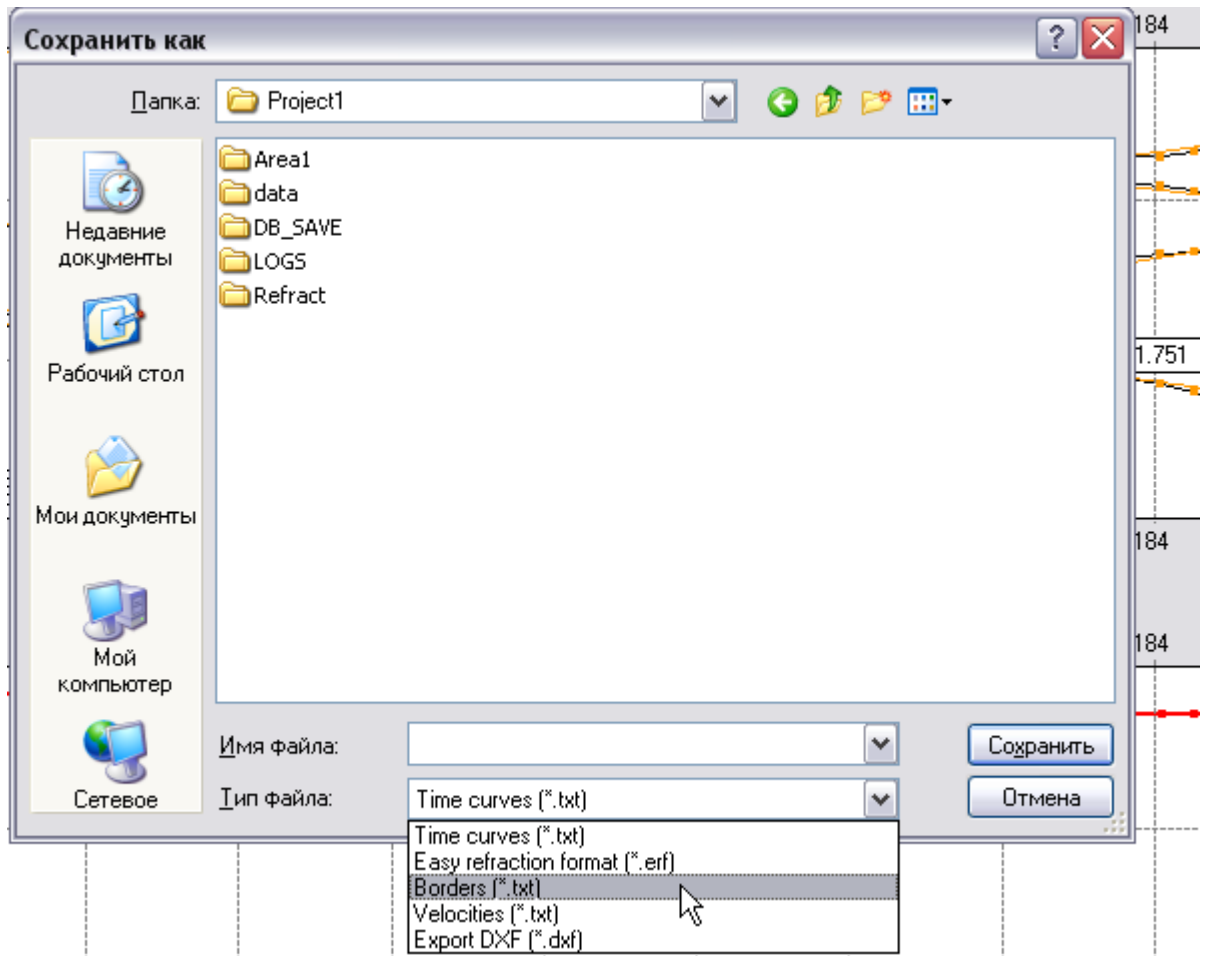


Then select the **Inversion/Reciprocal Method/Refraction Surface** menu item and specify the first layer velocity in the pop-up window to build a boundary between the first and the second layer using the t_0 travel time curve. The t_0 travel time curve needs to be selected with the left mouse button, and the residual travel time curve – with the right mouse button.

! To hide any of the layers, press **ctrl + layer number**.

Exporting the results

You can export the results of your work by selecting the **Data/Export** menu command. This will open the file saving dialog box. Select what to export and in what format to save it from the **File Type** drop-down list.



You can export the following data to a text (ASCII) file:

- X-coordinates, altitudes, velocities and layer depths as one ASCII table (**Plain text**)
- travel time curves (**Time curves**)
- refraction boundary depths (**Borders**)
- velocities (**Velocities**)

Refraction boundaries can also be exported in the DXF format.