

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

(Revised October 22, 2012)

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: <u>support@radexpro.ru</u> Website: <u>www.radexpro.com</u>

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: <u>http://www.radexpro.com/downloads/tutorials</u>

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.

Registered projects	
	New project
	Select project
	Remove from list
	Save list
roject directory:	Load list

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.

lew da	tabase		x
Title	My Project		_
🔽 Cr	eate subfolder		
	ОК	Cancel	

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.

X
New project
Select project
Remove from list
Save list
Load list

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.

C Rad	RadExPro 2011.1 >>> My Project				
Help	Options	Database	Tools	Exit	
 -	Create	new area			
	Create	new area			

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

ew area nai	me		×
My Area			
	ок	Cancel	
	ОК	Cancel	

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

🥂 Rad		1.1 >>> My		
Help	Options	Database	Tools	Exit
<u> </u>	My Are:			
		View map		
		Create line.		
		Rename Delete		

Enter the line name just like the area name.

ew line n	ame		
Line 1			
	ок	Cancel	

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.

RadExPro 2011.1 >>> My Project	
<u>H</u> elp <u>Options</u> <u>D</u> atabase Tools E <u>x</u> it	
<mark>My Area_Line 1</mark> ⊟ <mark>010 - data load</mark>	
	NEW RENAME
MB1 DblClick - Default action; MB2 - Context menu; MB1 - Drag flow to line to copy	TRASH

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.

SEG-Y Input	$\overline{\mathbf{X}}$
File(s) data\100.sgy data\115.sgy data\235.sgy data\335.sgy	Sample format Sample interval 1 Number of traces 0 Trace length 1024 Use trace weighting factor Big-enidan byte order (SEG-Y standard) Little-endian byte order Sorted by FFID:OFFSET Get all C Selection Sample interval 1 Number of traces 0 Trace length 1024 1024 C Selection Remap header value RECNO.41,,1817 SOURCE,41,,1857 ILINE_NO.41,,1897 XLINE,
Add Delete Load list Save list	Load remap Save remap
ОК	Cancel

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).

Select dataset	×
Object name	
	Location My Area Une 1 010 - data load
RenameDelete	Ok Cancel

Also, add the Screen Display module to the flow after the Trace Output module for monitoring purposes. The resulting flow should look like this:

My Project/My Area/Line 1/010 - data load	PROF LOCADO AND	Auto Autoc Autoc Autoc Autoc	
Help Options Database Tools Run Flow mode	Exit		
6EG-Y Input <- [multiple]	Towns forward	Deter langet	Data I/O
Frace Output -> line 1 - raw	Trace Input	Data Input	
Screen Display	Trace Output	Data Output	
	VSP Data Modeling	3D Data Input	
	3D Data Output	SEG-D Input	
	Super Gather	GSSI	
	Lamb: Solid Layer - Solid modeling	Load Text Trace	
	ЛОГИС	RAMAC/GPR	
	SCS-3 Input	SEG-2 Input	
	SEG-B Input	SEG-Y Input	
	SEG-Y Output	Text Output	
	2D Finite Difference Modeling		
	Ensemble Stack	Asymptotic CCP Binning	
			Deconvolution
	Deconvolution	Predictive Deconvolution	
	Custom Impulse Trace Transforms	Surface-Consistent Deconvolution	
	Nonstationary predictive deconvolution		
	DC Removal	Hilbert Transform	Signal Processing
	Resample	Amplitude Correction	
	Bandpass Filtering	Butterworth Filtering	
	Trace Math Transforms	Zero-Offset DeMultiple	
	Wave field subtraction	VSP SDC	
	wave field subtraction	V3P SDC	Interactive Tools
	VSP Display	Screen Display	Interactive roots
	3D View		
		3D Screen Display	
	Velocity Editor	QC Analysis	
	3D Gazer	Interactive Velocity Analysis	
	Plotting*	Advanced VSP Dispaly	
	VSP Migration	T-K Migration	Migration
	STOLT3D	Stolt F-K Migration	
	Curved Profile VSP Migration	2D-3D VSP Migration	
		20 30 Vol migration	
	Trace Math	X Interpolation	
	Trace Length	Trace Editing	
IB1 - Drag module: Ctrl+MB1 - Copy module: MB1 DbIClin	:k - Module Parameters; MB2 - Toggle module; Ctrl+MB2 DblC	Click - Delete	

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 – source coordinates (SOU_X) and receiver coordinates (REC_X) – to the seismic data. We will use **Near-Surface Geometry Input** module for that.

Create a new flow: **020 – Geometry Input**, add **Trace Input** module, select the **raw_data** dataset that we have created before and load all traces in the order they are stored in the dataset (for that chose **Get all** option of the **Trace Input**).

RadExPro 2011.3 >>> My project	TILATINATINATINATINATI	
<u>H</u> elp <u>O</u> ptions <u>D</u> atabase Tools E <u>x</u> it		
───My area─Line 1 = 010 Data load 020 Geometry Input		Â
Trace Input	×	
Data Sets raw_data Image: Constraint of the set	Sort Fields	REMAME TRASH
OK Cancel	Select from file File Database object Choose Get all	

Now we can add **Near-Surface Geometry Input** module to the flow. The module is dedicated to assigning geometry to seismic data acquired using different techniques typical for near-surface applications, including conventional refraction acquisition scheme.

When the module is added to the flow, you will see its parameter dialog as shown below. For assigning geometry to seismic refraction data select the **Refraction** tab of the dialog:

Reflection/MASW	Refraction				
			-		
	TryA	hand			
	1 2			12	
	T	Ţ	-	Ţ	
Receivers First Reciever Pos	ition 0	m i	Number Of Chann	els 12	1
Reciever Step	5	m			
Streamer Sources -					
Const Step	First Source Positi	on 5	m		
 Const Step Variable Step 	First Source Positi				
	First Source Positi Source Step	on 5	m		
	Source Step	5	m	r of	
C Variable Step			m Numbe	r of se Sources 3	
C Variable Step	Source Step Number of Forward Sources Source Nº Coor	5	m Numbe Revers	e Sources ³	
C Variable Step Offset Sources —	Source Step Number of Forward Sources Source Nº Coor 1 1	3	m Numbe Revers Sour 1	e Sources	[
C Variable Step Offset Sources — C Const Step	Source Step Number of Forward Sources Source Nº Coor	3	m Numbe Revers	ce Nº Coordinate]
C Variable Step Offset Sources — C Const Step Variable Step	Source Step	3 rdinate	m Numbe Revers Sour 1 2	e Sources 3 <u>ce № Coordinate</u> 1 2]

Our sample data was acquired according to the following scheme: our receiver streamer contained 48 geophones equally spaced every 1 meter. Shots on the streamer were made on channels 1, 12, 24, 36 and 48 (0, 11, 23, 35, 47 m accordingly). Offset shots we made 24 meters ahead of the 1st channel and 23 meters behind the last channel.

In case of refraction data, the module will calculate and assign shot point coordianates (SOU_X), receiver coordinates (REC_X), and source-receiver offsets (OFFSET) based on shot point numbers (FFID) and channel numbers (CHAN). For correct geometry assignment we will specify the following parameters:

Receivers:

- First receiver position 0
- Receiver step 1
- Number of channels 48

Streamer sources

Let us specify each source position manually. Select **Variable step** option to see the table with the shot point numbers and their coordinates. Set **Number of sources** to 5 and type their coordinates into the table: 1-0, 2-11, 3-23, 4-35, 5-47.

Offset sources

Select Variable step option, set Number of forward sources to 1 and type in the coordinate of -24 m.

The same way, set **Number of reverse sources** to 1 and type in the last shot point coordinate of 70 m.

! This data was acquired in such a way that the original field channel numbering is not sequential (1, 25, 2, 26, etc. – a result of some technical peculiarities of the receiving system). However, for correct geometry assignment we need the channels and shot points to be numbered sequentially, in the order they were acquired in field. To correct the numbering we switch on **Reassign FFID and CHAN headers** option. This would recalculate and reassign all shot point numbers and channel numbers according to the order the traces input the module in the flow.

Finally, the dialog shall look as shown below:

Reflection/MASW	Refraction		
	_		
***	Junt Junt		
	1 2	48	
	T T T	TTTTTTTTTTTTTTTTTTTTTTTTTTTTTTTTTTTTTT	
Receivers	1000 <u>00</u> 0		
First Reciever Po	sition 0 m N	lumber Of Channels	48
Reciever Step	1 m		
Streamer Sources	,	<u></u>	
C Const Step	Number of Sources 5	Source Nº	Coordinate 🔺
Variable Step		1 2	0
		3	23 *
Offset Sources —			
	Number of Forward Sources	Number of Reverse Source	es 1
C Const Step	Source Nº Coordinate	Source Nº	Coordinate
Variable Step	1 -24	1	70
Reassign FFID a	and CHAN trace headers		

When the parameters are set, click the OK to save your settings.

Now we will add Trace Output at the end of the flow to save the data with assigned geometry to a new dataset in the project database. We will call the new dataset geom_data and place it at the Line 1 level of the database (as shown below):

Select dataset Object <u>name</u> geometry_data	
Objects raw_data geometry_data	Location ⊡ My area ⊡ Line 1 010 Data load 020 Geometry Input
Rename Delete	Ok Cancel

The resulting flow shall look as following:

elp <u>O</u> ptions <u>D</u> atabase Tools Run Flo	w mode E <u>x</u> it		
race Input & raw data			Data I/O
race Input <- raw_data lear-Surface Geometry Input	Trace Input	Trace Output	
race Output -> geometry data	SEG-Y Input	SEG-Y Output	
race Output -> geometry_uata	SEG-D Input	RAMAC/GPR	
	SEG-B Input	логис	
	SEG-2 Input	GSSI	
	SCS-3 Input	Super Gather	
	Load Text Trace	Text Output	
	Data Input	Data Output	
	,	,	—Geometry/Headers
	Trace Header Math	Compute Line Length	
	Header<->Dataset Transfer	Header Averager	
	Header Output	Shift Header	
	Trace Header NMO/NMI	Surface-Consistent Calibration*	
		,	
	Screen Display	3D Gazer	
	Plotting*		
	DC Removal	Hilbert Transform	
	Resample	Amplitude Correction	
	Bandpass Filtering	Butterworth Filtering	
	Trace Math Transforms	5	
	·····,-···,		
	2D Spatial Filtering	Antenna Ringdown Removal	
	Burst Noise Removal	Ensemble Equalization	

Now click **Run** in the menu to execute the flow. As a result we will have a copy of the data with geometry assigned.

First arrival picking

Create a new flow named **030** – *Picking* 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.

Trace Input	×
Data Sets	Sour Fields
Add Delete	Add Delete Selection *:*
OK Cancel	O Select from file File O Database object Choose O Get all

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.

From t= 0.0 to 511.0 t Scale 10 Number of traces 200 X Scale 10 C WT //A C None Gain 0.3 Rotate C WT C WT C Individual Bias(%) 0 Variable spacing field Variable density display mode Normalizing factor Gain 2 Variable spacing field Variable density display mode Normalizing factor Gain 2 Variable spacing field Variable density display mode Normalizing factor Gain 2 Variable spacing field Variable density display mode Normalizing factor Gain 2 Variable spacing field C Ustom Define O None Bias(%) 0 Variable panels 1 O None Data/velocity Data/velocity Data/velocity Data/velocity Min.vel (m/s) 500.0 Max.vel (m/s) 1500.0 Max.vel (m/s) 1500.0 Max.vel (m/s) 1500.0	Display parameters	×
✓ Variable density display mode Normalizing factor Gain 2 Space to maximum ensemble width Image: Grey Image: Grey	Number of traces 200 TX Scale 10	O WI /VA O None Gain 0.3 O WT © Entire screen Bias(%) 0 O VA O Individual Show every 1
Axis Show headers C Display velocity Set velocity Min. vel (m/s) 500.0 Plot headers Header mark Min. vel (m/s) 1500.0	Space to maximum ensemble width Ensembles' gap Muliple panels 1	● Grey Normalizing factor Gain 2 ○ R/B ○ Entire screen Bias(%) 0 ○ Lustom Define ○ Individual 0 ○ Data/velocity ● Display data 0
Save Template Load Template Cancel	Plot headers Header mark	C Display velocity Set velocity Min.vel (m/s) 500.0 Max.vel (m/s) 1500.0

In the Screen Display dialog set the following parameters:

Click the **Axis** button and set the following axis parameters:

Primary lines	Time dt Values	Traces	Different dx Interval 10.0	Values 🔽
Secondary lines		field	C Multiple ′ C Different	V
Font size 20 Ok	Cancel	Margins Left axis margin	mm Topaxis margin 20 mr	n

Run the flow and see the data as shown below.



Select the **Tools/New pick** menu item and perform first arrival picking. After that you can pick first arrivals or extremums. Use zoom/unzoom buttons on the toolbar to select convenient scale.

You can select the mode of picking in the Picking parameters dialog achieved through the **Tools/Pick/Picking parameters** menu item.

Mode Manual C Hunt C Auto - Fill	Hunt opti	ation Test		al maximum le [,] elation windov		0.3
⊂ Linear Fill ⊂ Eraser		hunt direction v		- Hunt dire		
Parameters Peak C Zero:	Neg2Pos	Smoothing Window ler	10 C - 10 E	3 ро	oints	
C Trough C Zero: Guide window length	Pos2Neg 30	Drawing pa		ers Line style	La	abel picks
ОК	Cancel					

Picking can be done manually (**Mode: Manual**) or using on of the semi-automatic modes (**Auto fill** or **Hunt**). In the **Auto fill** mode the program automatically tracks selected event between two interpreter pick nodes according to the specified **Parameters**. In the **Hunt** mode it would try to follow the specified event in either one or both directions from the initial pick until it looses correlation.

To perform picking, click the left mouse button when the mouse cursor is over the selected point. An X mark will appear at that point showing a pick node. Click the left mouse button once more within the same trace to move the node to a new position, or click within another trace to place a new node. An erroneously placed node can be removed by right-mouse button double-click or moved to a new position by drag-and-drop with the right mouse button.



You need to pick all seismograms. To save the pick, select the **Tools/Save As...** menu item or right click on your pick in the **Pick List** window and select the same **Save As** command from the pop-up menu.

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**).

Save pick	×
Object name pick	
<u>O</u> bjects	Location
	⊡ · My Area ⊡ · Line 1 010 - data load 020 - pick 030 - refraction
C Save all C Save selection Rename Delete Pio	r 🔽 Append ok headers Ok Cancel

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

Pick headers			×
RECNO S_LINE SCDP SECOND SEGDGAIN SEQNO SFPIND SOU_CRL SOU_DATUM SOU_ELEV SOU_H2OD SOU_INL SOU_RESID SOU_SLOC SOU_SLOC SOU_STAT SOU_STAT1 SOU_STAT2 SOU_STAT3 SOU_X	▲ E	OFFSET PATH PICK1 PICK2 PREAMP B_LINE REC_CRL REC_DATUM REC_ELEV REC_H2OD REC_INL REC_RESID REC_SLOC REC_SLOC REC_STAT REC_STAT1 REC_STAT2 REC_STAT3 REC_UPHOLE REC_X	III
ОК	Cancel		

Working with the Easy Refraction module

Create a new flow named **004** – **Easy refraction** and add the **Easy Refraction** module to it. Select **Browse...** in the dialog box.

Choose Easy Refraction scheme	×
OK Cancel	Browse

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".

Choose Easy Refraction Scheme					
	Location ⊡ · My Area ⊡ · Line 1 				
Rename Delete	Ok Cancel				

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**.

Dialog	
⊡Ê My area ⊡Ê Line 1 Ê 010 Data load	My area\Line 1\pick
····- ¹ 020 Geometry Input ····- ¹ 030 Picking ····- ¹ 040 Easy refraction ·····O / pick	<>>>
	OK Cancel

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



You can make a number of manipulations with the time curves here: edit nodes, smooth them, interpolate, move etc. Refer to User Manual for the details.

As an example of such a manipulation we will mirror one of the curves. When we were picking fist breaks, the data from the first shot (at -24 m) was noted to be of very poor quality, so it was difficult to make a reliable pick. So now, if you have created a pick at the shot point -24, delete it (click of the pick in the list of picks in the left pane of the window and press **Delete** key on the keyboard). We will substitute this pick with a mirrored pick from the shot point at 70 m (although we need to understand that this is not 100% fair procedure, of course). For that, switch off all time curves except of the one from shot point 70 m, select it with the left mouse button and select Time Curves/Mirror curve. As a result we will have a symmetrical time curve tied to SP -23 m:



Time curves count: 6 X: 11,09 m T: 74 ms



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 or press 1 on the keyboard. You will see a pink circle of the marker of the fist layer. You can change its diameter by rotating the mouse wheel while keeping Shift key pressed.

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 `key on your keyboard) to uncheck it.

Automatic travel time inversion

With the **Easy Refraction** you can invert travel time curves either automatically, or manually. We will check both methods below.

For automatic inversion (prior layer selection with the marker is required) select **Inversion/Automatic inversion** menu command (you can also press F5 or click on the green arrow on the toolbar).



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. Velocities above and below the boundary are colorcoded.



Press t key to show/hide layer velocity values at the section.

Manual travel time inversion

If necessary, all refraction processing stages can be carried out manually with the full controll over the intermediate results.

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/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. The result is shown below:



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** menu and select **Reciprocal time Ieveling**. 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 t0 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/Velocity analysis and time depth functions** menu. The result is shown below:



Finally, you need to select the velocities of the upper layer and specify parameters of velocity calculation in the lower layer.

V1 Estimation	V2 Estimation			
Automatic 🔽		Automatic 🔽		
Value 500	Window width	15		
	Window step	10		
Correct values manually	Correct value	Correct values manually		

Select the Inversion/Refraction Surface menu item to see the following dialog:

V1 Estimation – you can specify a constant velocity of the upper layer or make it calculated automatically by linear interpolation between the values estimated at every SP.

V2 Estimation – parameters how to estimate the lower layer velocities from the time depth function. Here **Window width** specify how many nodes of the time curve will be taken into account for velocity estimation at every window position (defined by **Window step**).

We will keep the defaults here and click the OK to see the following result:



Exporting the results

You can export the results of your work by selecting the **File/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.

ĺ	Сохранить как				? 🔀 184
	<u>П</u> апка:	🚞 Project1	 3 3 	• 🖽 🥙	
	Недавние документы Рабочий стол Мои документы	Area1 data DB_SAVE LOGS Refract			1.751
	Сетевое	<u>И</u> мя файла: <u>Т</u> ип файла:	Time curves (*.txt)		аранить тмена
			Time curves (*.txt) Easy refraction format (*.erf) Borders (*.txt) Velocities (*.txt) Export DXF (*.dxf)		

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.