

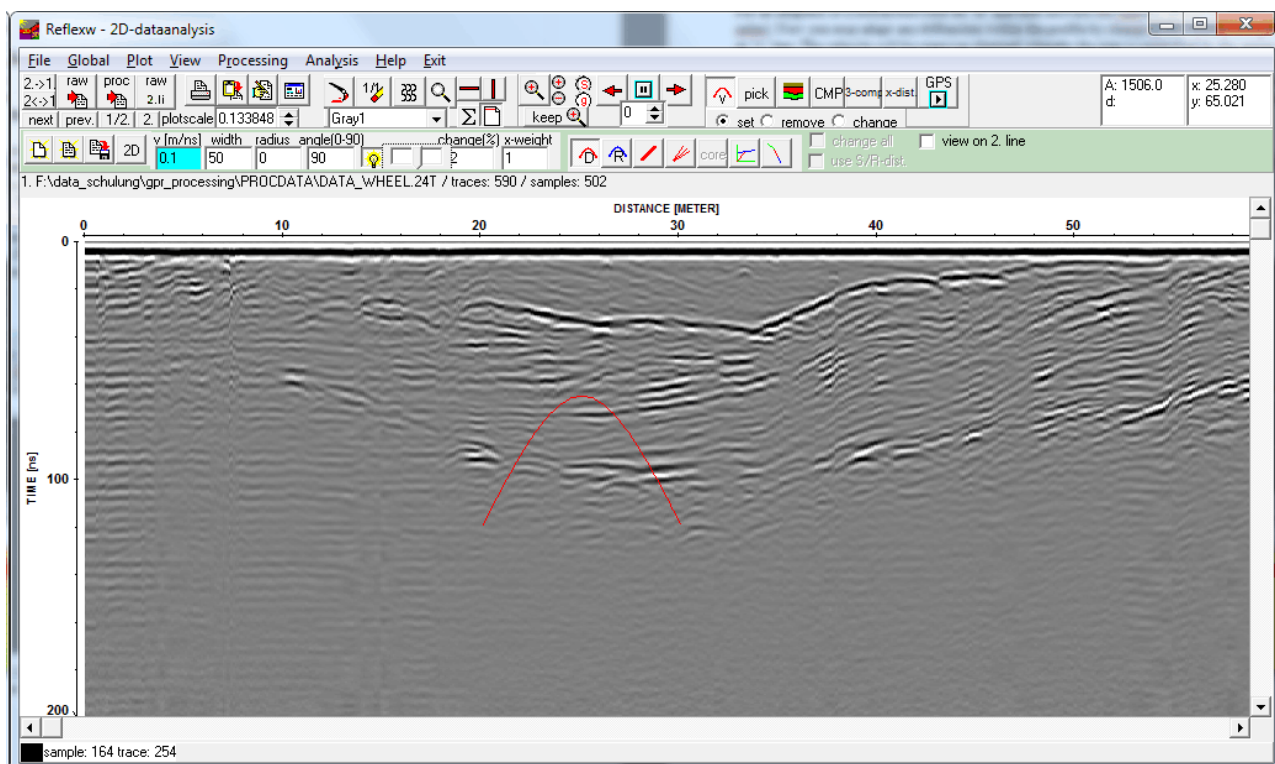
Introduction to the interpretation of ZO-data within REFLEXW

In the following the different interpretation tools for Zero offset data (GPR, seismic reflection, acoustic) are presented. Chapter I describes the different possibilities of analysing velocities, in chapter II the picking of reflectors or individual elements is discussed.

I. Velocity analysis

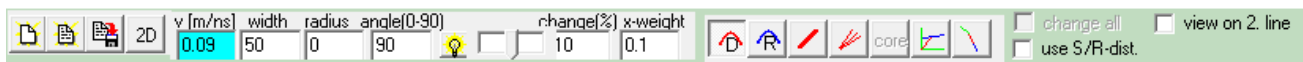
I.1 interactive velocity analysis

The interactive velocity analysis within the 2D-Dataanalysis allows the interactive adaptation of **diffraction** or **reflexion hyperbola** by calculated hyperbola of defined velocity and width. There is also the possibility of fitting **linear features** either by interactively changing a line or by setting two points. The speed button **core** allows to interactively vary the velocities of the single layers of the individual cores. The velocities may be stored on file and may be reloaded at any time. The velocities are combined into a 2D-model by using a special interpolation (see also option x-weight). Such a 2D-velocity distribution may be used in a subsequent step for the migration or the depth conversion.



Click on image in order to view an animation concerning the use of the interactive velocity analysis

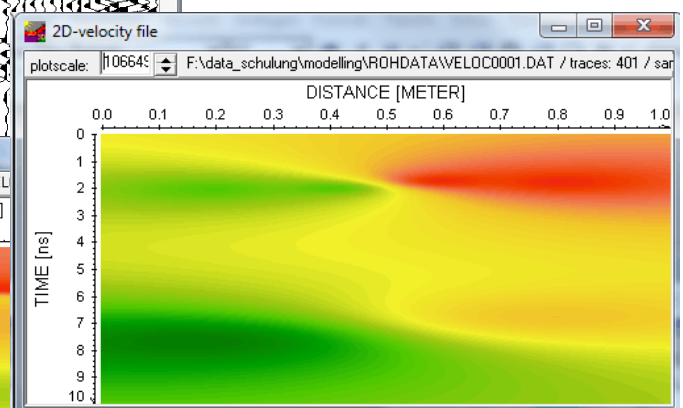
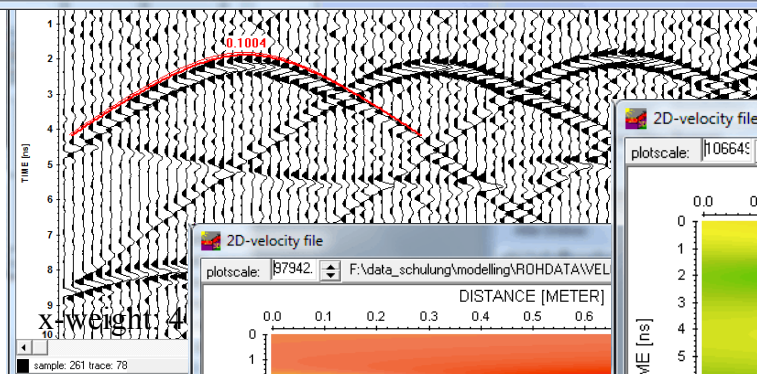
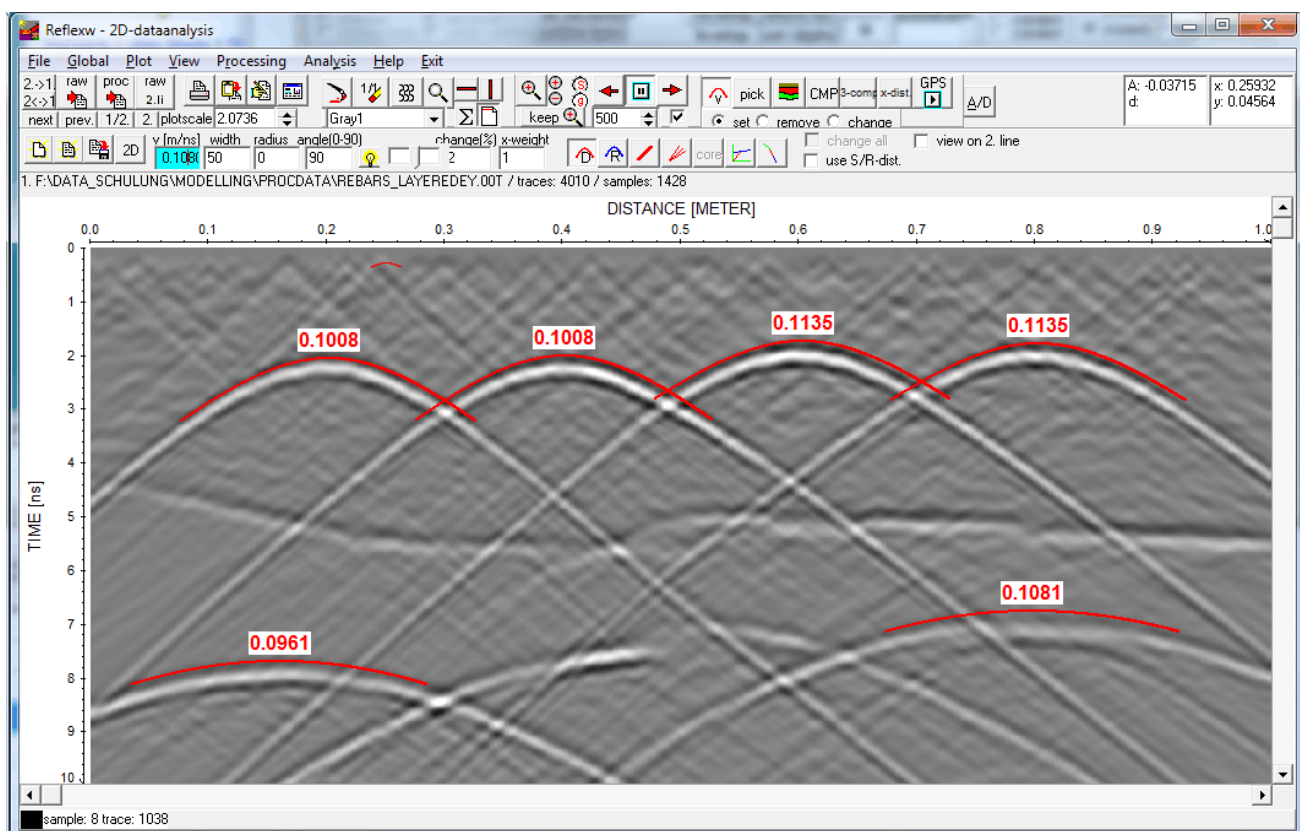
I.1.1 diffraction adaptation



For an adaption of a diffraction click on 'D' and then activate the v[m/ns] option (it appears in blue color). Now you may adapt any diffraction within the profile by changing the velocity using the '<' or '>' key. The velocity will be stepwise changed whereby the step is controlled by the option change (%). The width of the calculated hyperbola is given by width in traces.

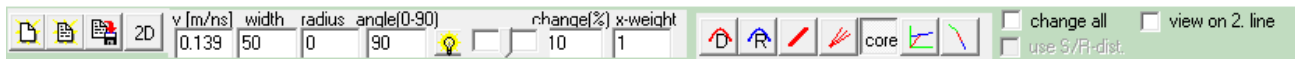
The adaptation must always be done for the phase which corresponds to time zero, this means if time zero has been set to the first arrival, the adaptation of the hyperbola must also be based on the first arrivals because the curvatures along the signal length (in time) remains the same.

The option 2D allows to display the 2-dimensional velocity distribution of the chosen adaptations. The interpolation is controlled by the parameter x-weight (increase x-weight if a more layered distribution is wanted).



x-weight: 1

I.1.2 core data adaptation



For an adaption of coredata first a core datafile must be loaded. This is done using the option view/core data. The coredata file must be created outside Reflexw. The file is an ASCII-file which contains the following informations:

1. line: CORE + any identification
 2. line: current location (x,y and z) within the 2D-line (floating point format)
 - 3.-x. line: layernumber thickness quality-factor mean-velocity layer-velocity
- example of 3 cores (velocities are given):

CORE 10

10.0000	0.0000	0.0000
1 1.220000	1 0.140	0.140
2 1.550000	1 0.134	0.130
4 0.660700	1 0.125	0.098

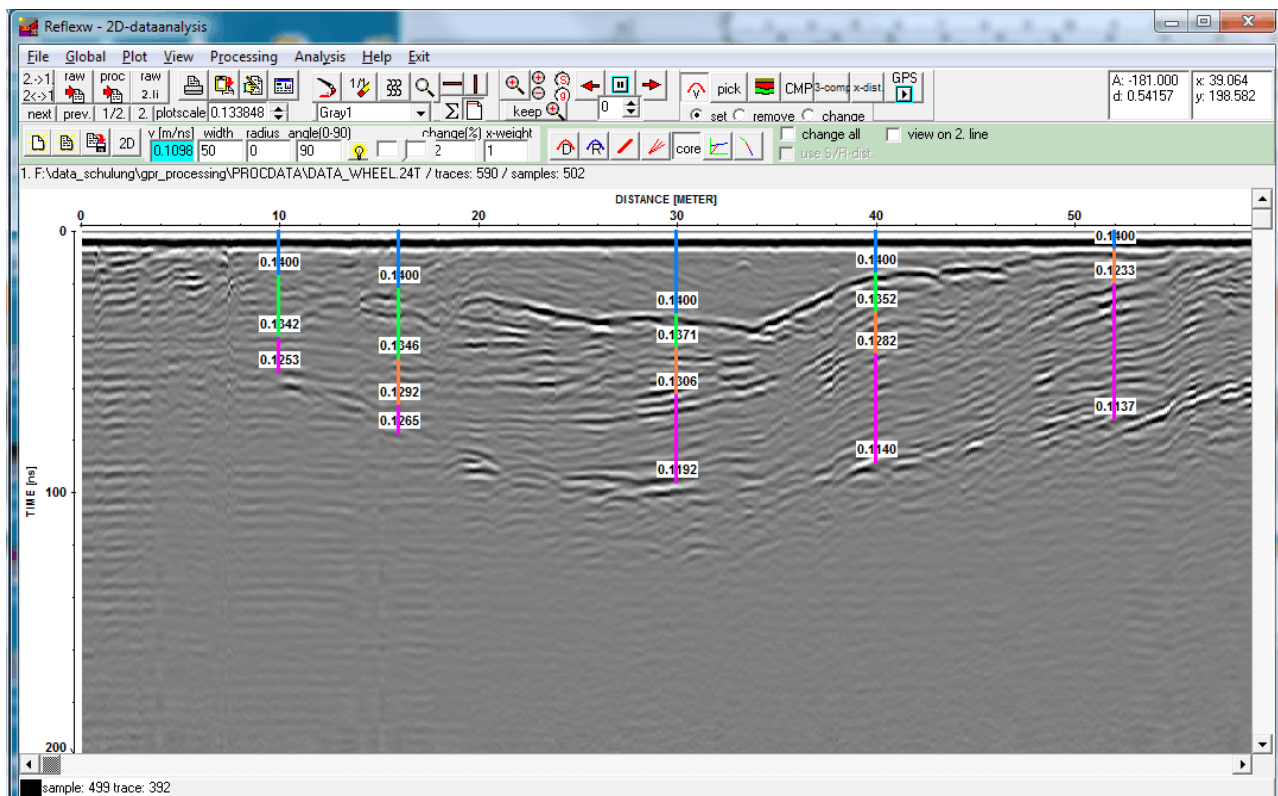
CORE 16

16.0000	0.0000	0.0000
1 1.580000	1 0.140	0.140
2 1.750000	1 0.135	0.130
3 1.000000	1 0.129	0.114
4 0.610700	1 0.126	0.110

CORE 30

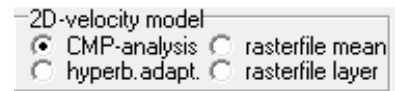
30.0000	0.0000	0.0000
1 2.240300	1 0.140	0.140
2 0.860000	1 0.137	0.130
3 1.000000	1 0.131	0.114
4 1.660700	1 0.119	0.098

If the core datafile does not contain the velocities of the individual core layers all velocities are preset to the current adaptation velocity. With the speed button core activated you may interactively adapt the core bars to layer boundaries visible within the 2D data. Click on any layer bar which you want to adapt. The fill style of this layer bar changes. With the adaptation parameter v highlighted it is now possible to change the velocity of this distinct layer by pressing the key < and > respectively or using the mouse wheel (see below). With the option **change all** activated all layer velocities of the actual cores are changed simultaneously.

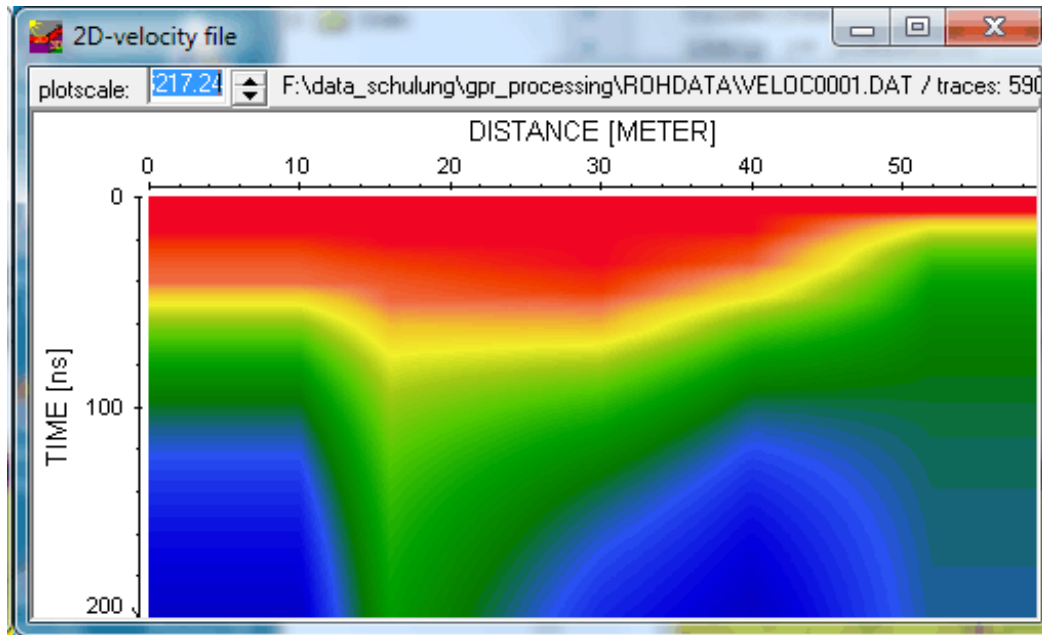


New in version 8.3: When saving the adaption a vla-velocity file as well as 1D-VEL files (used within the CMP-velocity analysis module) and a 2DM file containing all 1D-VEL-files are automatically created.

The VLA-file may be used for the layershow (see next chapter) and the 2DM-file may be used for a time-depth conversion or a migration based on 2D-velocities using the option CMP-analysis.



The option 2D again allows to display the 2-dimensional velocity distribution of the chosen adaptations. Now the interpolation is done based on the layers defined by the cores. The velocity distribution is stored under the file veloc001.dat under rohdata.



II. Picking

Picking may be used for defining a layer boundary or single elements. It also may be used for some processing steps like static or topographic correction.

Different picking possibilities are available:

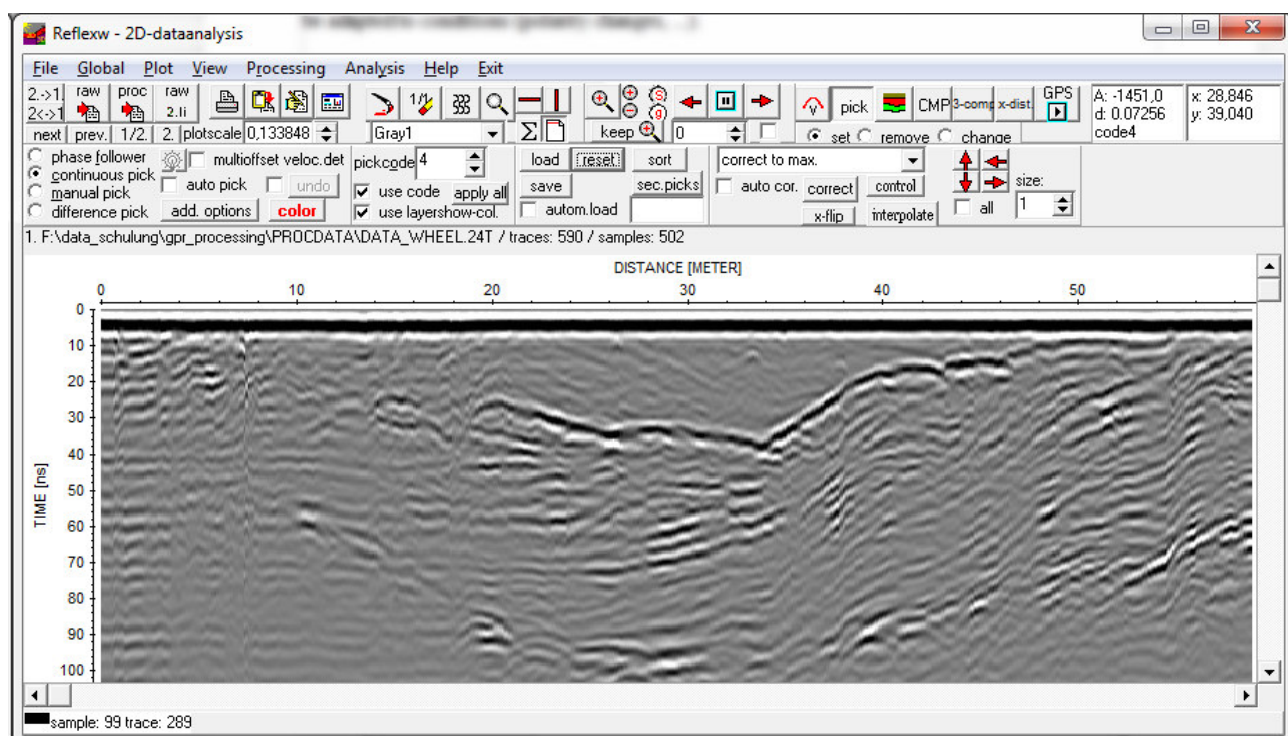
phase follower: use this option for long profiles (plotoption PixelsPerTrace recommended) and for the case that the reflector is quite continuous. The automatic phase follower is stopped using the right mouse button or the escape key. Then you may force the phasefollower to continue the picking at a distinct place. Already existing picks are replaced by the new ones - for a detailed description please see online help.

continuous pick: use this option for a continuous manual picking using pressed left mouse button, it might also be useful for continuously changing or removing existing picks.

manual pick: use this option for single trace picking or for changing individual picks

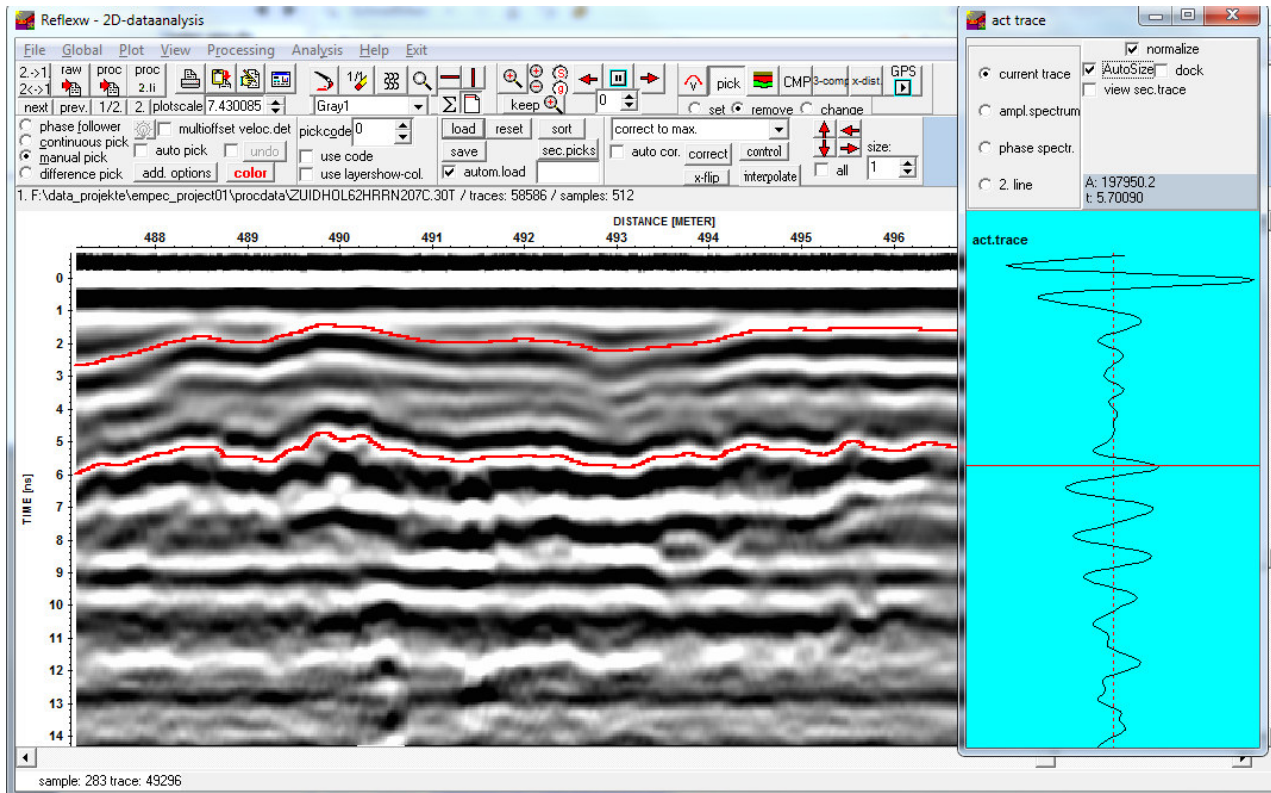
difference pick: With this option activated the travelttime difference between two successive picks is automatically determined

auto pick: Within this menu you have the possibility to automatically pick the wanted onsets. Depending on the needs you may choose between single objects like small diffractions, first arrivals(e.g. from refraction seismics) or continuous reflector



Click on the image in order to view an animation about the use of the phase follower. The automatic picking is stopped by pressing the escape key when the phase has been lost and you may force the follower to pick the correct one by clicking on it. The use of the continuous picking allows to smoothe the transitions zones afterwards.

To be considered: it is important to pick the correct phase depending on the used time zero. In the lower picture the time zero has been set to the positive maximum of the first arrival. It is assumed that the two picked reflections have the same polarity as the first arrival because in all cases the reflection is based on a decrease of the velocity of the lower layer (first arrival is the air/surface reflection in this case). Therefore again the positive maxima have been picked. Of course this must be adapted to conditions (polarity changes, ...).



II.1 Picking distinct reflectors

In the following the picking of different reflectors within REFLEXW is described including the different layer discrimination possibilities.

II.1.2 pick a distinct reflector

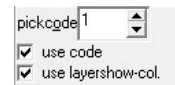
There are two possibilities for the discrimination of the different picked layers.

The first one is to pick each reflector separately, save the picks for this layer and combine the pick files together within the layershow (see chap. II.1.3).

The second one is the use of the pick codes. These codes are stored with each pick and may also be used for a discrimination. The picks are saved within one file and an ASCII-export might be used in order to generate a text file for all picks for the use within a further interpretation tool.

It is not necessary that the reflector is continuous but it also may consist of several broken parts. In the following the different steps are described:

1. enter the module **2D-dataanalysis**
2. **load** the wanted GPR or seismic ZO(zero offset) profile - comment: GPR profiles are normally ZO-lines
3. activate the option **pick**
4. **pick the reflector** using one of the possible picking possibilities (see chap. II). Use the pick code corresponding to the layer number if the discrimination of the layers shall be done based on these codes.



first option - using the layershow:

5a. use the option **save** for saving the picks of one reflector using the format Reflex Win. Enter the following parameters:

global code: any code

layer number: number of the layer - necessary for the following combination of the different picked reflectors within the layer show, min. layer number is 1, max. layer number is 100. Each reflector should receive an individual number.

mean or layer velocity: enter a mean or a layer velocity for the time-depth conversion (see also chap. II.1.3.1).

filename: filename for the picks (automatic extension is pck)

6a. **reset** all picks if you want to pick a new reflector

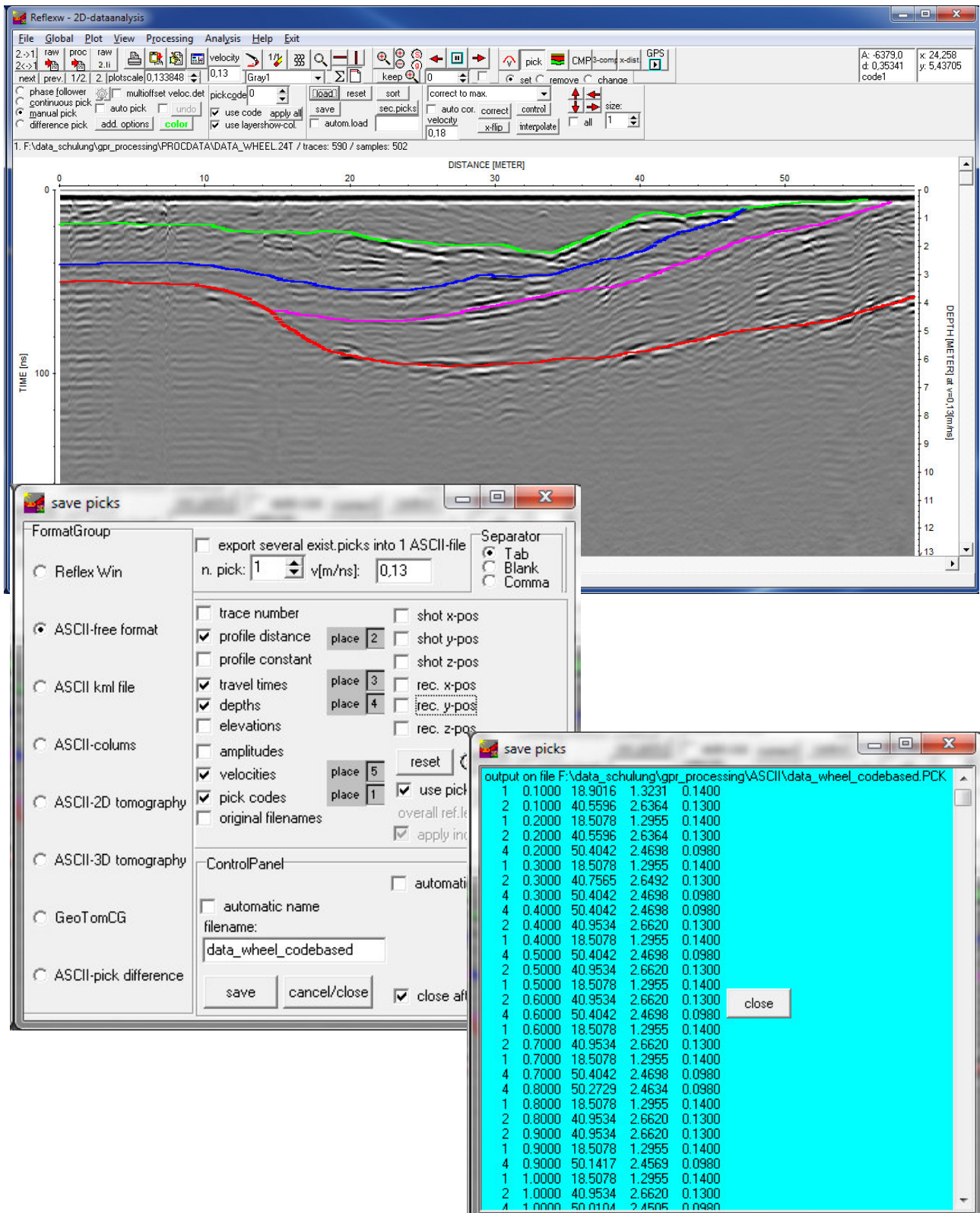
7a. **repeat** step 4-6a until all reflectors are picked - to be considered: the layer number must be changed for every different reflector. The max. layer number is 100.

Second option - using the pick codes:


5b. Increase the pickcode and pick the new layer. With each pick the traveltime, distance, traceheader coordinate, pick code and a velocity are stored. The velocity represents a mean velocity and not a layer velocity.

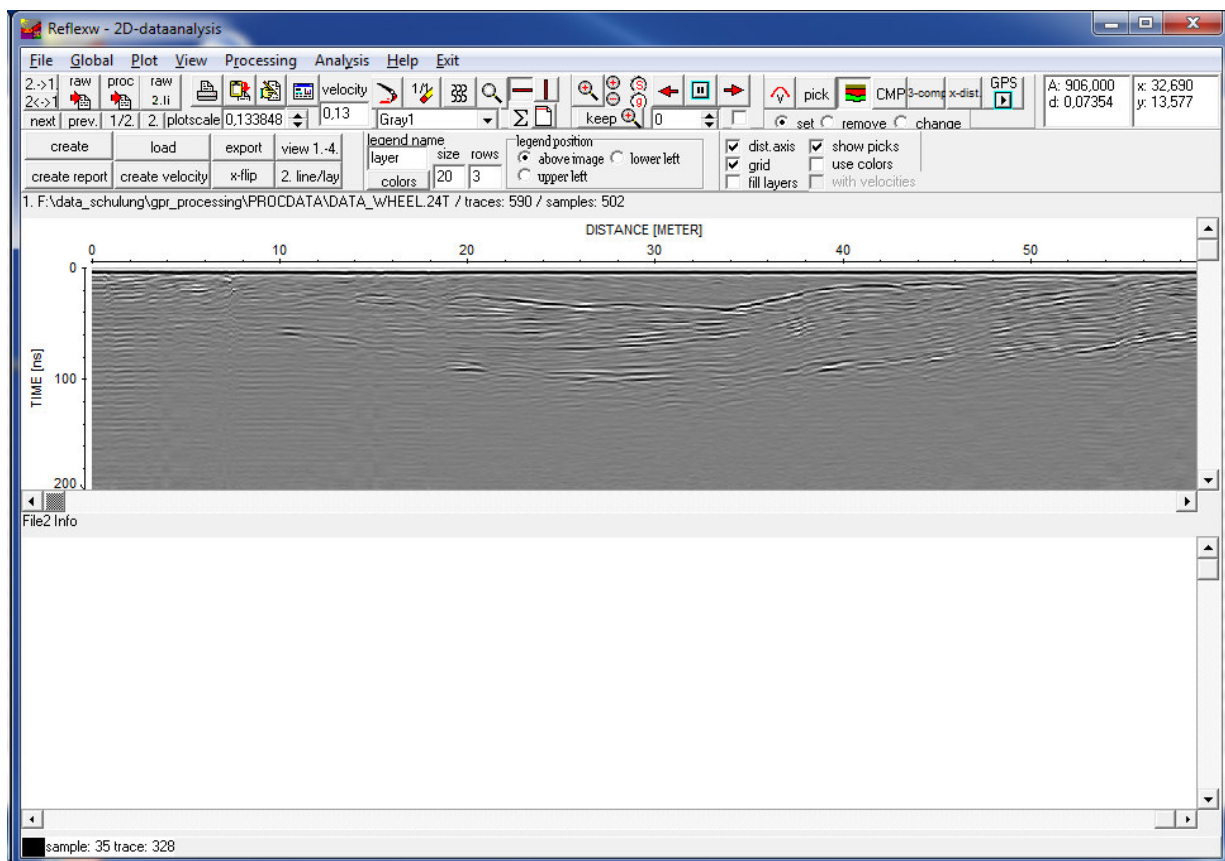
6b. **repeat** step 5b until all reflectors are picked.

7b. Use the option **save** for saving the picks of all reflectors using the format Reflex Win or any of the ASCII-formats. The global code and layernumber and the velocities are of no interest.



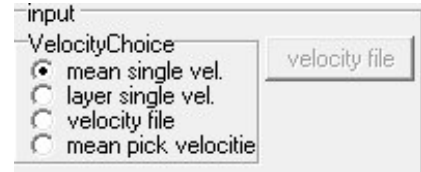
II.1.3. create a layershow

1. pick a distinct reflector - it is not necessary that the reflector is continuous but it also may consist of different broken parts (chap. II.1.2)
2. save the picks of one reflector using a special name (e.g. layer 1)- enter the layer number, the mean velocity and a layer code
3. repeat step 1 and 2 until all reflectors have been picked
4. enter the layer-show
5. create a velocity file if layer velocities shall be used instead of mean velocities (chap. II.1.3.1)
6. create the layer-show (chap. II.1.3.2)
7. create a report (chap. II.1.3.3)
8. activate the option **layer-show**  or under analyse/layer-show - the corresponding ZO-profile must be loaded - the new layershow parameter panel opens.



II.1.3.1 generate a velocity file for the time-depth conversion (open layershow panel)

For the **time-depth conversion** using the option create LayerShow (see section II.1.3.2) it is necessary to take a **velocity** into account. You have the choice between mean single velocity, layer single velocity, velocity file and mean pick velocities.



Choosing "**mean single vel.**" means that the mean velocity specified at the saving of the picks of the corresponding layer (see also SAVE under PICK) is used for the time-depth conversion. Only one value for the complete layer is possible. The velocity is given as the mean velocity and not as the layer velocity. The time-depth conversion of each layer boundary is independent from the conversion of the upper boundary.

Choosing "**layer single vel.**" means that the velocity specified at the saving of the picks of the corresponding layer (see also SAVE under PICK) is assumed to be a layer velocity and it is used for the time-depth conversion. Only one value for the complete layer is possible. The velocity is given as the layer velocity. Therefore the time-depth conversion of each layer boundary depends on the conversion of the upper boundary (see also option interpolate layer). The result is the same as using the option velocity file with a constant velocity for each layer (see below).

Choosing "**mean pick velocities**" means that the mean velocities stored with the picks are used for the time-depth conversion. The velocities have been created e.g. using the option multioffset veloc.det within the pick-menu or by setting the velocities manually. The velocities are given as the mean velocities and not as the layer velocities. The time-depth conversion of each layer boundary is independent from the conversion of the upper boundary.

If you want to use any of these three options you may skip this chapter and you may continue with chapter II.1.3.2.

The second possibility is to use an **ASCII velocity datafile** (e.g. test.vla). In that case, the velocities for each layer may vary laterally and they are given as layer velocities in contrast to the mean pick velocities. Such an ASCII-file contains the following information:

comment line

comment line

1.velocity line

....

n. velocity line

Each velocity line contains:

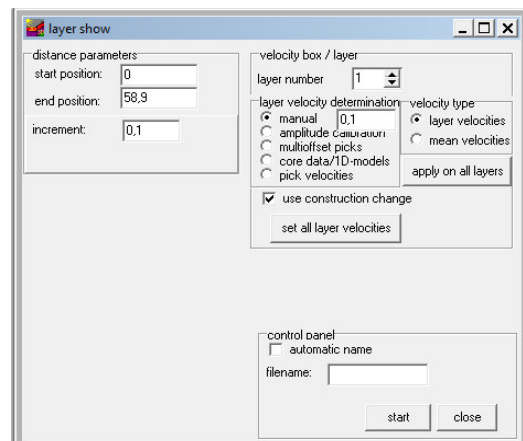
distance v-layer1 v-layer2 v-layer10 or v-layer 100

The layer velocities may not be defined at each desired distance position but the program performs a linear interpolation between the given values.

Such an ASCII velocity datafile can either be generated manually using any editor or using **core files** (see chapter 6.1.7.4 within the manual) or using the option **create velocity** in the LayerShow panel. The latter option is described below in detail:

1. activate the option **create velocity** within the layershow panel - the create velocity menu opens (see figure on the right).

2. there are five different ways to generate an ASCII velocity datafile: The layer velocities are set **manually**, which implies no lateral change of the velocity in one layer, or the velocities are calculated using an



amplitude calibration of the picked layer boundaries, or the velocities are calculated based on different **multioffset picks** or they are taken from **core data** or taken from the pick velocities. The second case (amplitude calibration) is only integrated within REFLEXW for completeness. We do not recommend this method (see manual, chap. 1.12.3.4). You may use different ways for the different layers.

3. Activate the option **manual** and enter the **velocities** for the individual layers and the **filename** for the velocity file (the velocity file automatically has the extension vla).
4. activate the option **start** for the generation of the ASCII file. The file is shown in a window and activate the option **close** for closing the display of the file.
5. activate the option **close** for closing the create velocity menu.
6. The ASCII-file can be changed manually using any editor.

II.1.3.2. combine the different picked reflectors into one layer model (open layershow panel)

This option offers the possibility to combine individual pick files stored on file, to plot them together with the wiggle-files and to output them in report form on printer or file. The maximum number of combined picks per layer boundary is equal to the max. number of traces per profile. The maximum number of layer boundaries is 10 (until ver. 2.1) or 100 (from ver. 2.5).

The picked traveltimes of the reflectors are transformed into depths using either a mean velocity for a distinct layer or based on a laterally varying velocity file.

In the following the individual steps of creating a layer-show are described.

1. activate the option **create** within the layershow panel - the create layer show menu opens (see figure on the right).

2. activate the option **pick files** within the layer show menu and choose all wanted pickfiles (multiple choice using the shift or str-key) - the choosen pick files are listed in the textbox on the right.

3. Control the other parameters like start/end position, increment and the velocity choice for the time-depth conversion.

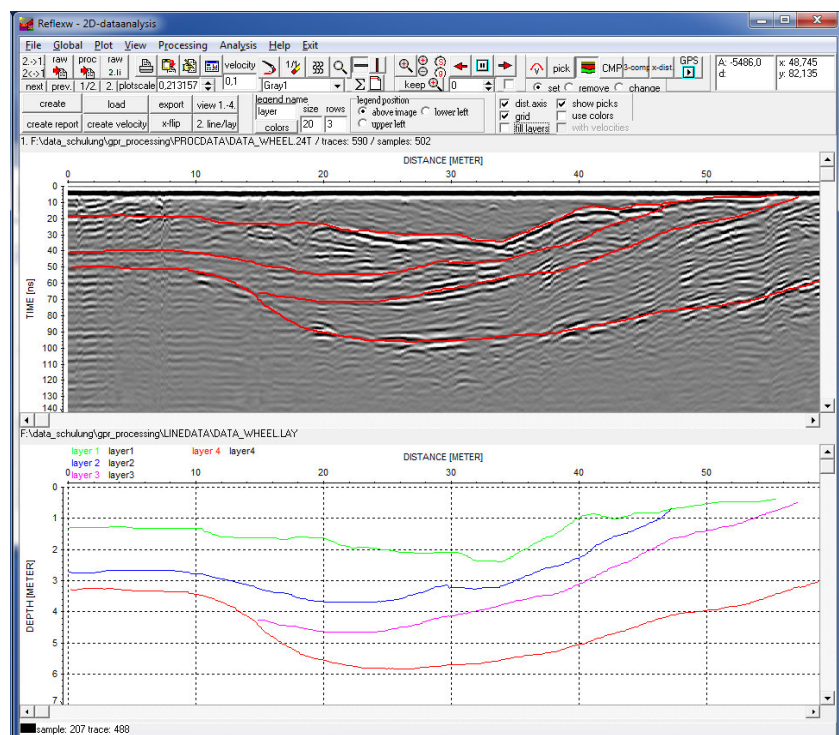
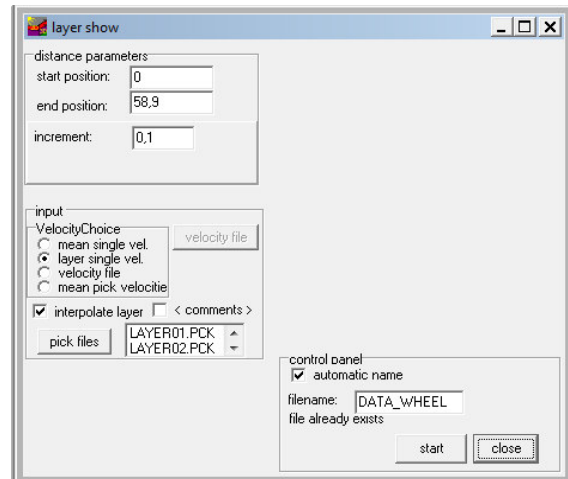
If you want to use a **velocity file** for the time-depth conversion instead of the **mean or layer pick velocity** stored with each pick file first you must create such a velocity file (see section II.1.3.1). Then you must activate the option velocity file and load the wanted velocity file. The option **interpolate layer** controls how the special case of broken reflectors are handled (see below).

4. Enter a **filename** for the layer-show (extension is automatically lay) and activate the option **start** in order to create the layer-show.

5. activate the option **close** for closing the create layershowy menu.

6. the layer-show is displayed in the lower window - the traveltime picks are displayed together with the ZO-profile within the upper window.

Note: If the option depth axis within the plotoptions is activated the depths for the picks of the upper window are only comparable to the depths of the layers in the lower picture if mean velocity has been used for creating the layershow and if all mean velocities of the picks are equal to the velocity for the depthaxis display.



7. Any layershow can be reloaded using the option **load** within the layershow panel.

To be considered: If the **mean single velocity** or **mean pick velocities** are used for the **time-depth conversion**, the time-depth conversion of each layer boundary is independent from the conversion of the upper boundary.

In contrast, if an **layer single velocity** or an **ASCII velocity datafile** is used, the time-depth conversion of each layer boundary depends on the conversion of the upper boundary. Therefore, in that case one has to control how the time-depth conversion for a special layer is done if any upper layer is not continuously picked. This is done using the option **interpolate layer**.

If this option is activated every upper layer is assumed to be continuous even if it is not picked and either an interpolation or an extrapolation both for the layer points and the velocities is done. Based on these calculated values the time-depth conversion of the actual layer is done.

If the option is deactivated, the velocities of the next upper picked layer are used. This might result in a sharp step of the depth of the actual layer at the point where any of the upper layers is broken.

II.1.3.2. generate an output report of the layer-show (open layershow panel)

The results of the current layershow can be stored in a report. The output is performed either on a printer or on a file with an arbitrary name. According to the individual settings, the report contains information about single layers like e.g. thickness, velocity, amplitudes and coordinates. In addition, actually loaded coredata and marker comments can be reported. It is also possible to reduce the information on single points e.g. to represent only material depth between two construction changes.

1. activate the option **create report** within the layershow panel - the create report menu opens (see figure on the right). Precondition: a layershow must be loaded or just created.

2. enter the necessary parameters for the output:
- start/end position and increment
- layers: enter the numbers of the layers for the report output: e.g. 1-5 or 1-5,8,9

3. activate the option **start** in order to start the report output.

II.2 Picking single objects within REFLEXW

II.2.1. Picking hyperbolas along one profile line

The following flow described one way how to mark single objects (hyperbolas) and to save the parameters x, z and ID on an ASCII file for a further interpretation with any other CAD-system.

1. enter the module 2D-dataanalysis
2. load the wanted GPR or seismic ZO(zero offset) profile
3. activate the option **pick**
4. activate the option **manual pick**
5. enter the ID of the hyperbola within the paramter **pickcode**
6. click on the cusp of the hyperbola within the profile using the left mouse button
7. repeat step 5-6 until all hyperbola cusps have been picked
8. enter the option **save** and activate the pickformat **ASCII-columns**. In addition you also may save the picks using the REFLEXW-Win format in order to load the picks at a later stage again.
9. Activate the options depths, amplitudes and pick codes.
10. enter the wanted velocity in m/ns for converting the 2-way traveltimes into depths
11. enter the wanted filename and press save
12. each line of the output file contains the following informations:
profiledistance profileconstant-start traveltime depth amplitude code

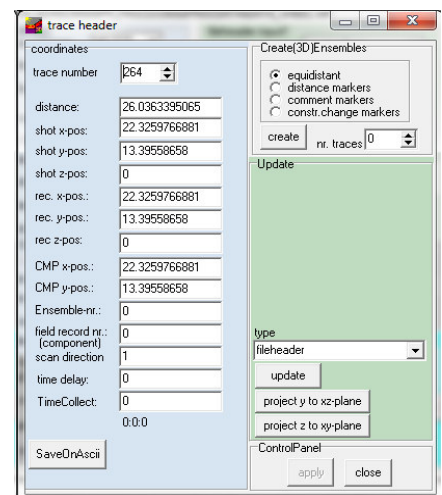
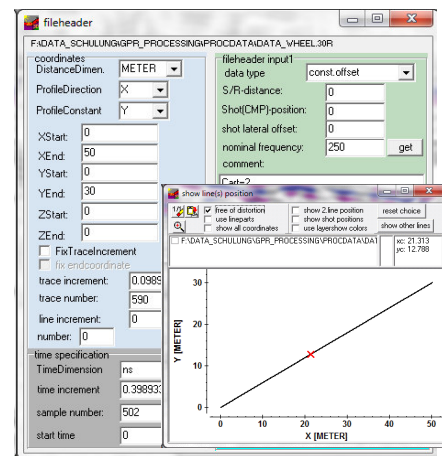
II.2.2. Use of xy-coordinates

II.2.2.1 Use of xy-coordinates defined within the traceheader for one single profile

The ASCII-output described above does not contain the real xy-coordinates but the distance coordinates along the profile together with the start coordinate in the profile constant direction. Therefore these coordinates only correspond to the real xy-coordinates if the profiledirection is strictly orientated into x- or y-direction. If this does not hold true (e.g. diagonal lines) you must take into account the so called traceheader coordinates in addition. These coordinates may represent relative or absolute (GPS) coordinates. The relative coordinates may be generated from the overall fileheader coordinates if the lines are straight. The GPS-coordinates may be loaded from differently formatted ASCII-files. Reflexw supports GPS-datafiles from most of the GPR-systems. In addition time based GPS-data may be imported if the GPR or seismic data also contain time informations.

A. Define the traceheader coordinates from the fileheader:

1. Define the geometry of your file within the **fileheader** either during the **import** or within the edit fileheader menu (option **edit fileheader**). The example shows a diagonal line.
2. After having defined the fileheader coordinates (do not forget so save them before when you have changed the fileheader coordinates within the fileheader menu) activate the option **ShowTraceHeader**.
3. The traceheader menu opens. Here you may actualize the traceheadercoordinates either based on the fileheader coordinates or based on different ASCII-files (see also guide for GPS-coordinates). Enter within the Actualize panel for **type** fileheader and press the button **actualize**.
4. Now the traceheader coordinates are defined based on the given fileheader coordinates. You may check the coordinates using the **trace number** edit button.



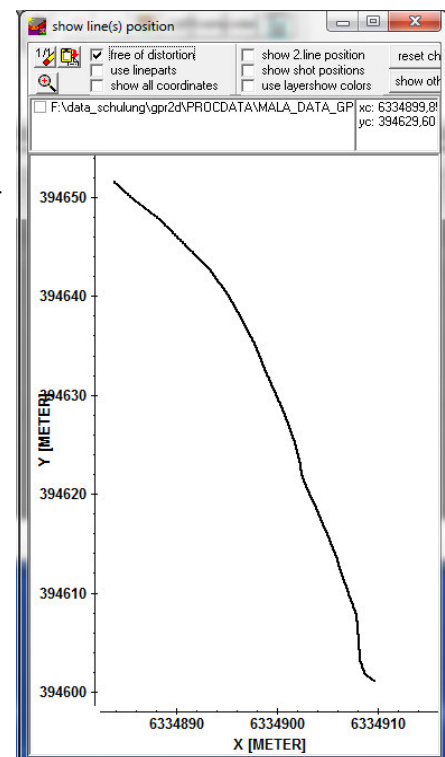
B. Define the traceheader coordinates using GPS-coordinates:

If GPS-data are simultaneously acquired it is possible to synchronize the GPR-data with the GPS-coordinates. The original GPR data may either be time based or based on a wheel. The different GPR-systems use different synchronization types. Mala, Utsi, IDS or PulseEkko for example generate a gps-file which contains both the tracenummer of the GPR-file and the GPS-coordinates. For most data acquisitions systems it is possible to automatically import the gps-data into the Reflexw file during the import and to perform a subsequent UTM-conversion. A linear interpolation will be automatically done where no GPS-data are present. The option calculate distances sums up the distance along the gps-line and stores it into the Reflexw traceheader. The GPS-coordinates may be controlled and edited within the edit traceheader tabella.

trace-nr.	distance	shot-x	shot-y	shot-z	rec.-x	rec.-y	rec.-z	time delay	gain	time collect
218	13.091933415	394613.25191	6334905.8295	-0.512333333	394613.25191	6334905.8295	-0.512333333	0	1	0
219	13.360163136	394613.51005	6334905.7570	-0.510166666	394613.51005	6334905.7570	-0.510166666	0	1	0
220	13.628392861	394613.76816	6334905.6841	-0.508	394613.76816	6334905.6841	-0.508	0	1	0
221	13.792059890	394613.91952	6334905.6216	-0.504470588	394613.91952	6334905.6216	-0.504470588	0	1	0
222	13.955726917	394614.07086	6334905.5595	-0.500941176	394614.07086	6334905.5595	-0.500941176	0	1	0
223	14.119393944	394614.22220	6334905.4972	-0.497411764	394614.22220	6334905.4972	-0.497411764	0	1	0
224	14.283060967	394614.37355	6334905.4346	-0.493882352	394614.37355	6334905.4346	-0.493882352	0	1	0
225	14.446727991	394614.52485	6334905.3725	-0.490352941	394614.52485	6334905.3725	-0.490352941	0	1	0
226	14.610395015	394614.67623	6334905.3102	-0.486823529	394614.67623	6334905.3102	-0.486823529	0	1	0
227	14.774062036	394614.82757	6334905.2475	-0.483294117	394614.82757	6334905.2475	-0.483294117	0	1	0
228	14.937729055	394614.97891	6334905.1856	-0.479764705	394614.97891	6334905.1856	-0.479764705	0	1	0
229	15.101396076	394615.13025	6334905.1233	-0.476235294	394615.13025	6334905.1233	-0.476235294	0	1	0

Now the traceheader coordinates are defined based on the given fileheader coordinates. You may check the coordinates using the option profile line (trace header coord.) under view.

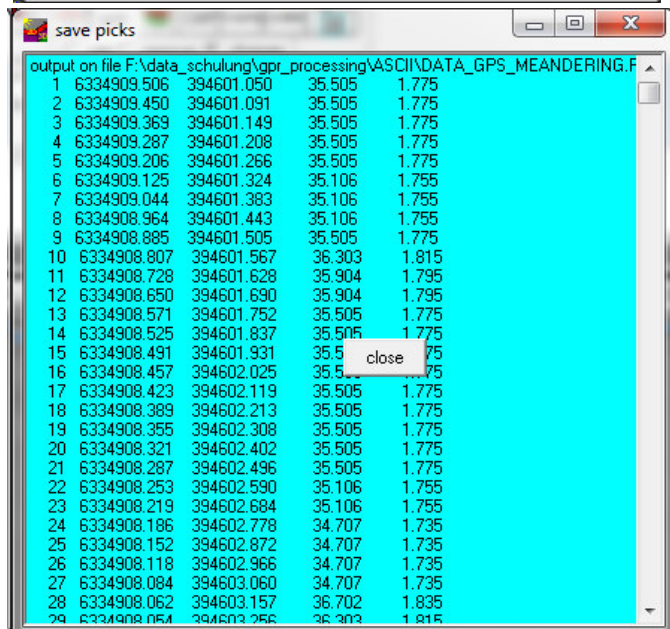
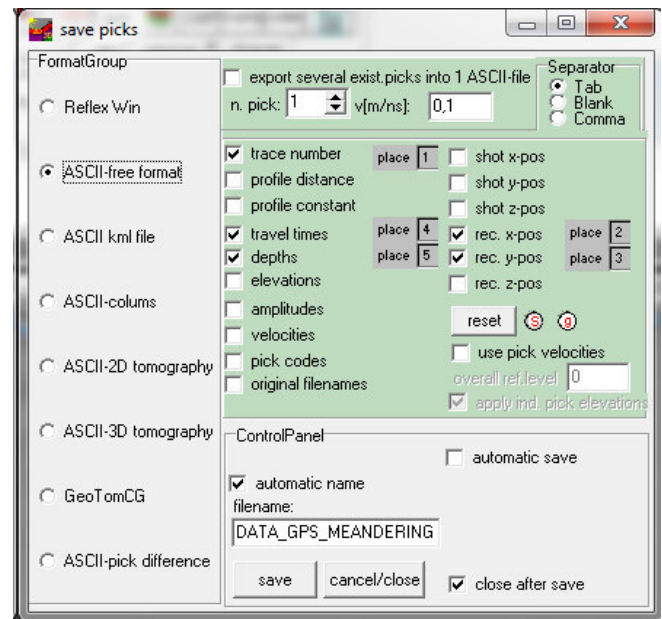
A subsequent processing step named **make equidist.traces** under processing/ TraceInterpolation/ Resorting may be used in addition to interpolate the non-equidistant data in such a way that the resulting data are equidistant. The non-equidistant data are resampled in x-direction based on the filter parameter trace incr. and the distance values stored in the individual trace headers of each trace. In addition the start distance and the end distance (starting and ending position of the new profile) have to be specified in the given distance dimension. By default the start distance and the end distance are determined from the traceheaders. By the manual input you may extract a distinct part from the profile.



Picking:

1. pick the hyperbolas like described in chapter 1.
2. When saving the picks using the format ASCII-free format you may activate for example the options trace number, rec. x- and y-pos, travel times and depths.
3. 12. Now each line of the output file contains the following informations:
`tracenumber rec.x-pos rec.y-pos traveltime depth`

For zero offset data the shot and the receiver coordinates are the same and you only may consider one coordinate pair.



II.2.2.2 Use of xy-coordinates for several profiles

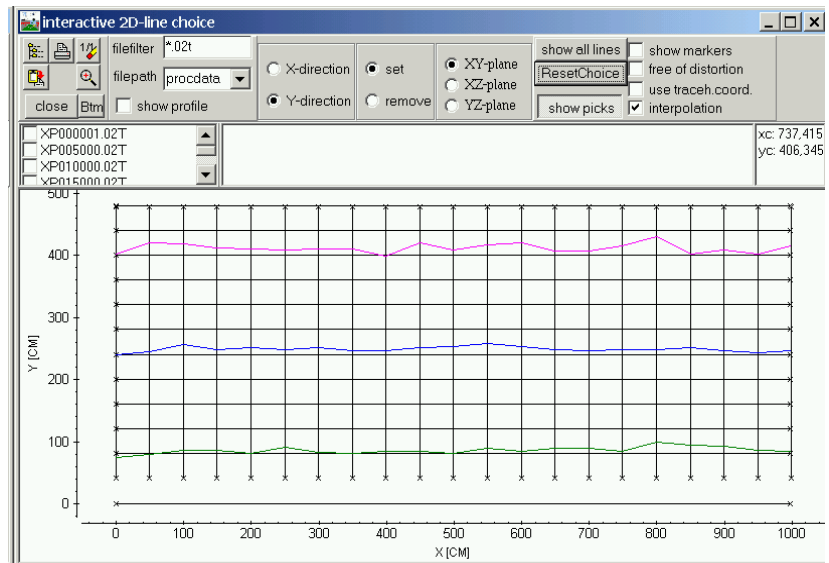
It is also possible to save the picks of different 2D-profiles into one ASCII-file.

1. For that purpose first you must save the picks of the 2D-profiles into different Reflexw-pickfiles using the pick format Reflex Win.

2. You may control the location of the picks within the interactive choice menu - enter for that purpose file/interactive choice within the 2D-datananalysis - the interactive 2D-line choice menu opens

3. Choose the wanted filepath and press show all lines

4. Click on show picks and choose the wanted pickfiles (multifile choice using the shift or strg key)



5. Picks belonging to the same code are connected if the option interpolation is activated. If the option use code within the 2D-dataanalysis is activated the layershow-colors are used for the display of the picks.

6. If the picks are correct you may leave the interactive choice menu and enter the pick save menu in order to save all the picks for the different 2D-lines into one single ASCII-file.

7. Within the save pick menu you must choose the “ASCII-colums” or “ASCII free format” pick format and you have to activate the option “export several existing picks into 1 ASCII-file”.

8. Click on save and choose all wanted ReflexWin formatted pickfiles (multi filechoice using the shift or strg key).

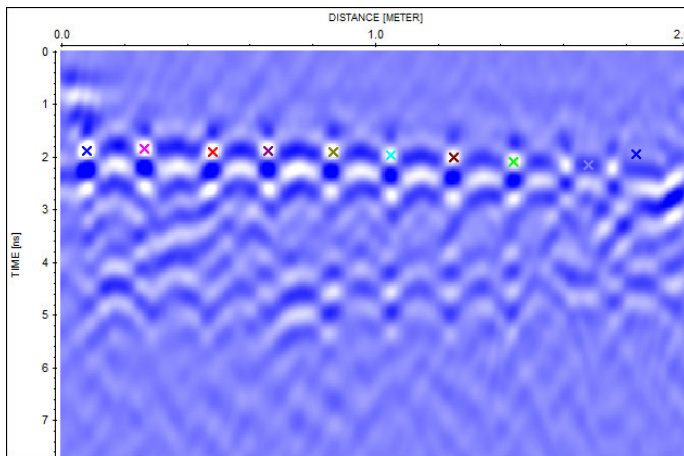
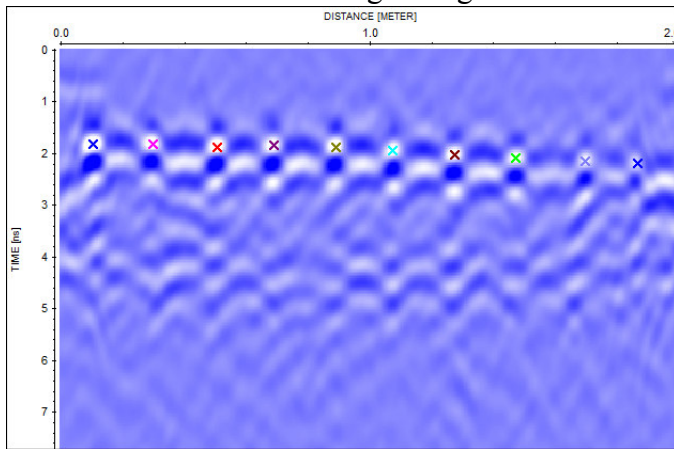
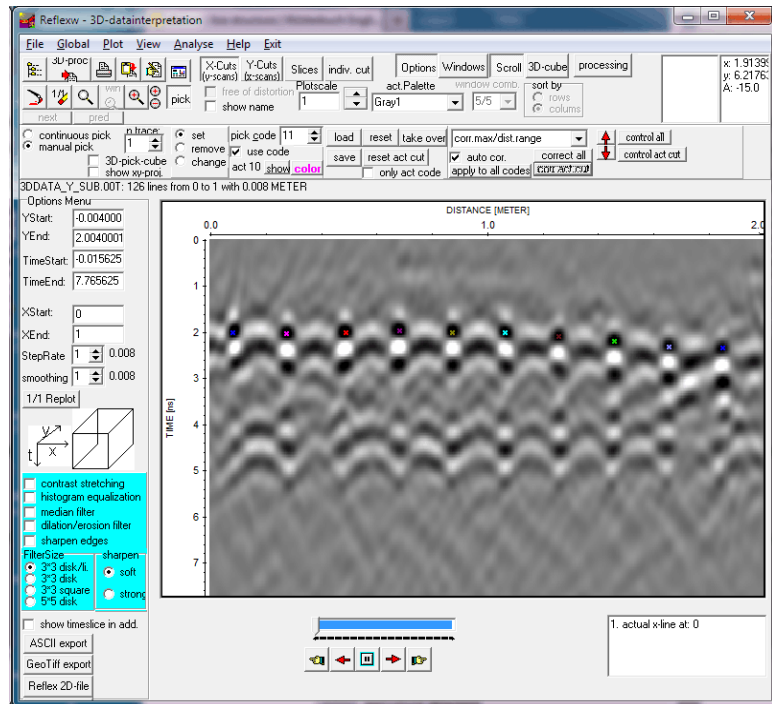
9. Each line of the resulting ASCII-file contains for example the following values:
traveltime depth amplitude x-shot y-shot z-shot x-receiver y-receiver z-receiver code(optional)
Depths and amplitudes are set to zero if the corresponding options depth and amplitudes are deactivated.

II.2.3 Picking linear structures within a 3D-datafile

Reflexw allows a fast picking of single objects within the 3D-datainterpretation.

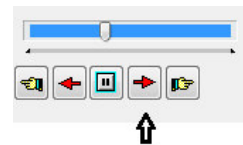
The following procedure may be applied to all types of single objects like pipes, rebars or other line structures.

1. Load a 3D-file within the 3D-datainterpretation and choose Scroll and the appropriate cut
2. Activate pick and activate use code and auto cor. based on corr.max/dist.range (option only act. code is deactivated). The parameters for the corr.max/dist.range may be entered within the global settings menu (parameter pickcorrect traces which should be set to 1 for data with a small cut increment).
3. Click on apply to all codes
4. Pick all elements of one block (e.g. one set of rebars orientated in one direction) using different codes (the F4 key may be used in order to increase the code) within the actual cut
5. Go to the next cut using the right



scroll button (hand)

6. Click on take over - the picks from the previous cut will be taken over to the actual cut
7. Click on the right arrow in order to scroll through the complete 3D-block. The new picks will be displayed. You can stop the scrolling at any time.



8. Control the picks by scrolling back. If any element will be lost due to a lateral jump for example (see example on the left for the two last picks) you may manually change the false pick:

- enter the code of the false pick (here 9)
- activate the option only act code
- change the correction type if necessary
- activate change and click on the correct pick position
- activate the option take over and scroll using the right red arrow to the end of the block, the wrong picks will be changed according to the new settings.

9. Make the same for all wrong picks (item 8)

10. Save the picks. It is recommended to store all different picks sets on different files as this

simplifies the semi automatic picking.

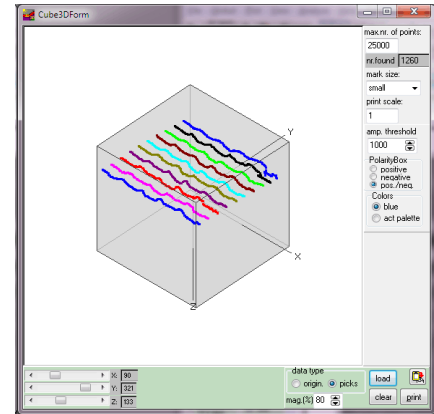
11. You can check the picks using the option view/fast data cube display. Activate picks and click on load.

12. Deactivate the option only act code and reset all picks using the option reset

13. Enter a new code for the starting pick of the next set of picks, e.g. the lower picks. Make sure that the new pick code(s) has not been used before.

14. Perform step 4 to 11 to the next set of picks.

15. Perform step 12 to 14 for all wanted pick sets.



After having picked and saved all wanted structures it is possible to view the picks within the 3D-cube display.

Enter for this purpose the option view picks surfaces for 3D-cube and choose the pick files. Different display options are available, e.g. for the color or the shading.

