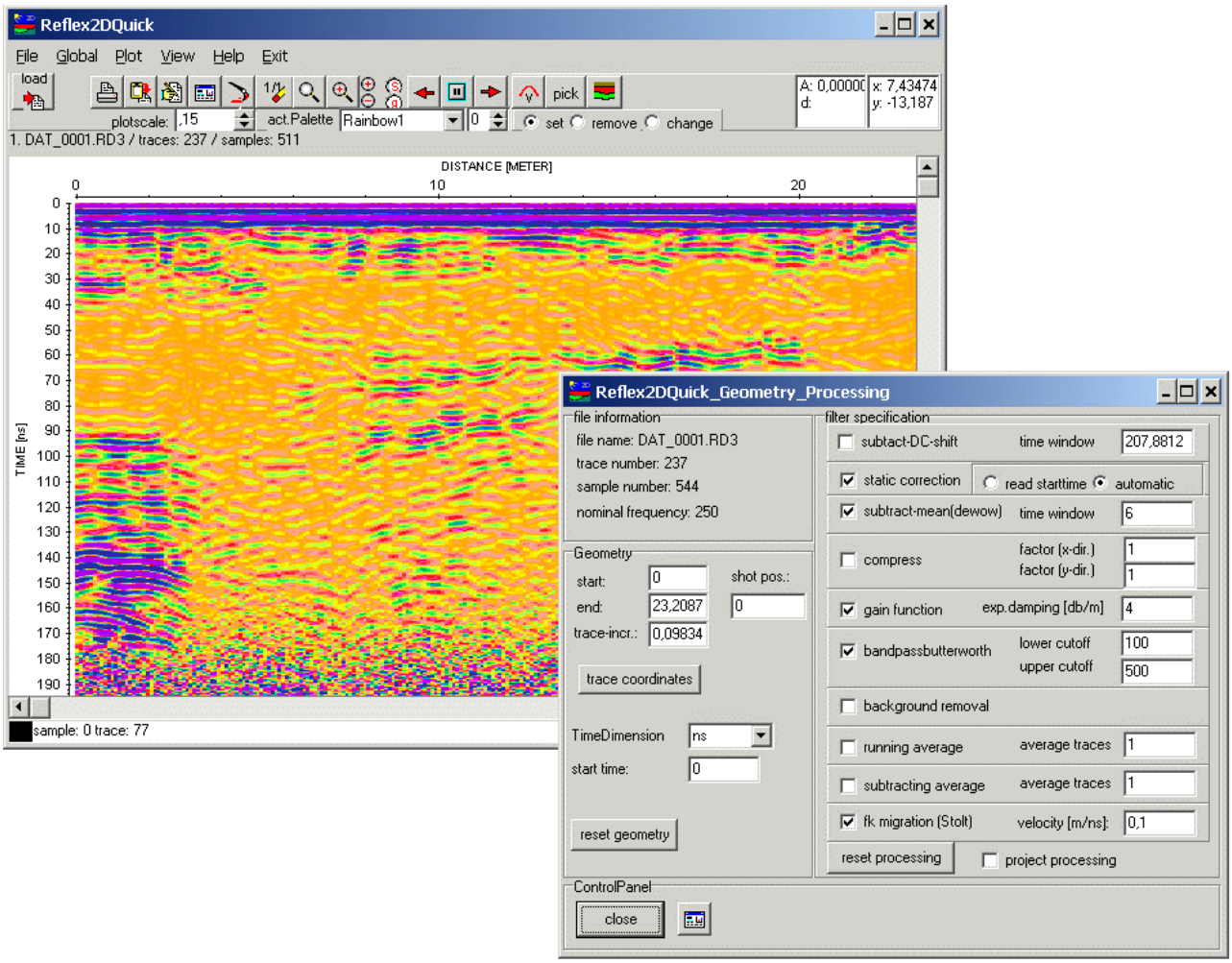
Reflex 2D-Quick

The program **Reflex 2D-Quick** allows an easy import, display, processing and interpretation of 2-dimensional **G**round**P**enetrating**R**adar zero offset and **seismic** single shot data. The program offers the following possibilities:

- direct import from different formats (SEGY, SEG2, Mala, Gssi, PulseEkko, Utsi, Ids see chap. 1)
- easy change of the distance scaling and the geometry (chap. 2)
- standard processing with predefined default parameters which can be manually changed (chap. 2)
- different display possibilities like point or wiggle mode, scaling and zooming functions (chap. 3)
- printing the profiles with free scaling (chap. 4)
- interactive velocity adaptation for zero-offset, single shot (reflection and refraction) or VSP data (chap. 5)
- picking the onsets and combine the picked phases to a layer model (chap. 6)
- export the data to other formats like SEGY, SEG2 (chap. 7)



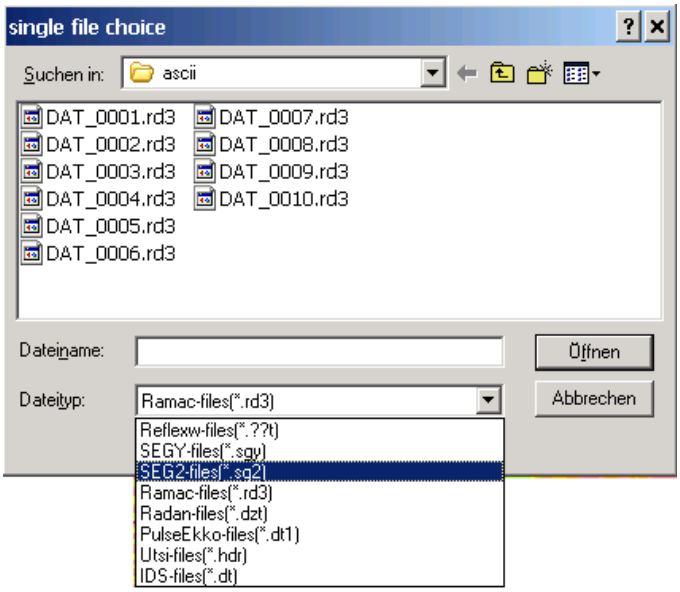
The program is useful for the following applications:

- GPR or seismic reflection constant (zero) offset data
- GPR or seismic single shot data (e.g. refraction seismics or single CMP's)

contents

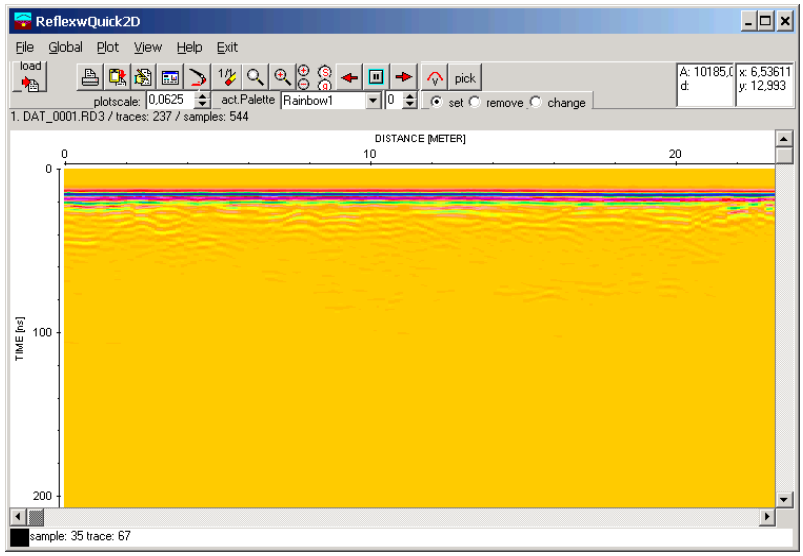
1. load and display the data

When pressing the button load or file/load the filechoice menu opens. Here you may specify the datatype of your original data:
SEGY-files *.sgy
SEG2-files *.sg2
Ramac-files *.rd3
Radan-files *.dzt
PulseEkko-files *.dt1
Utsi-files *.hdr
IDS-files *.dt
REFLEXW *dat



The data are plotted without applying any filter and using a distance axis. If available within the original data the traceincrement is read in. If not the default value 1 is assumed.

It is possible to start the program twice for a comparison of two different datasets. Consequently, you are also able to compare a processed dataset with the original dataset.



The following options are available for displaying, printing and analysing the data:

load: load the original data

print the current data files (see also Print Menu, chap. 4).With the option showmarker activated (see PlotOption) the markers are printed in addition. The size of the markers are taken from the size defined within the symbol font (see FontSettings). With the option Pick activated the current picks are printed in addition. The size of the picks are taken from the size defined within the symbol font.

copy to clipboard

enter Geometry_Processing menu (chap. 2)

enter PlotOptions (chap. 3)


scroll to the left


stop scrolling


scroll to the right

The spin button located beneath the autoscroll options defines a delay factor in order to slow down the scrolling. Value 0 no delay - value 50 max. delay.


replot current line with current zoom parameters


resets the x- and y-scale values (zoomvalues) to 1 and replots the current line


enable magnifying glass function


enable manual zoom - With the option ZOOM an arbitrary area of the data set can be selected and plotted in full screen size.The area to be enlarged, a rectangle, has to lie within a data set. Pressing the left mouse button you determine a corner of this rectangle and by moving the mouse with pressed button the desired area is opened.
The zoom range may be changed step by step using the small + or - buttons on the right side. Clicking on the + button increases the zoom by 10 %, clicking on the - button decreases the zoom by 10 %, this means the x-y-range to be shown will be larger.
The current zoom may be stored using the small "s" button and may be restored using the small "g" button.


enters the velocity adaptation MenuItem

pick button: enter the Pick Menu (chap. 6)


enter the LayerShow (chap. 6)

act.palette: load the wanted color palette from the stored palettes (see also PlotOptions menu).

plotscale: enter multiplication factor for the color-amplitude assignment or enter multiplication factor for the wiggle size. The minimum and maximum amplitude values are controlled by the multiplication factor Amplitudescale. With a value of 1 for Amplitudescale the amplitudes range from -2048 to 2048 for unnormalized data and from -1 to 1 for tracenormalized data. With a Value of 0.0625 the amplitudes range from -32768 to 32768 (see also Amplitudescale under Pointmodeattributes).

1.1 File MenuItem

From this menu you may open a file, export the data to different formats, copy the current image to the clipboard or print out the data.

Load: load the original data (see chap. above).

Print: Use this option if you want to print out the data (chap. 4). With the option showmarker activated (see PlotOptions) the markers are printed in addition. The size of the markers are taken from the size defined within the symbol font (see FontSettings). With the option Pick activated the current picks are printed in addition. The size of the picks are taken from the size defined within the symbol font.

Export: allows the conversion of REFLEX-formatted data into different export formats (e.g. SEGY) - see also Export Menu (chap. 7).

Enter Geometry_Processing menu: enter Geometry_Processing menu (chap. 2)

Edit TraceHeader: opens the trace header menu (chap. 2.2).

Processing flow: displays the processing flow of the current file.

CopyToClipboard/File: copy the current windows into the clipboard (suboption clipboard) or into a file (suboption file). Different file-formats like BMP, JPEG or TIFF are supported.

Image1ToClipboard: copy the window into the clipboard.

Exit: leave the program

1.2 Global MenuItem

printersetup: enters the printersetup menu.

info: enters the info windows about REFLEX 2D-Quick.

fonts: enters the font menu for either the text fonts or the number fonts. Please use only true type fonts because only these fonts are able to be rotated.

cursor: specify the wanted cursor.

global settings: this menu allows to specify some global settings parameters (see also global settings, chap. 1.2.1)

language: choose the language for the automatic labelling of the axis.

1.2.1 Global settings

this menu allows to specify some global settings parameters:

delay value: dependent on the computer used the display of the hyperbolas or the continuous picks, e.g., might be significantly delayed. This is due to the need of the computing time intensive refreshing of the screen. This problem can be resolved by entering a delay value n for the display of the hyperbolas, the continuous picking or the phase follower (value n = 1 - no delay). Then, only every nth call the hyperbolas for example are shown. The delay value n must be individually set based on the computer used. If the display of the hyperbolas or the continuous picks is still delayed with the chosen delay value n, please increase n. The current setting is stored within the ini-file when leaving the program.

ScrollSize: enter the fracture part of the image to be scrolled by one scroll step. For example ScrollSize 1/10 means that scrolling is done with a step rate of 10 % of the total size of the image. ½ means a step rate of 50 %. The new scrollsize is only available after a new zoom range has been entered. The current setting is stored within the ini-file when leaving the program. After having changed the scroll size you must reset a possible zooming using the option reset and afterwards reenable the zoom range in order to activate the new scrollsize.

complete plot by scrolling: if activated the data are completely replotted when scrolling in horizontal direction. Otherwise only the scroll part is replotted. Activate this option is the data are not clear when scrolling.

Check disk space: activate this option if you want that the program controls the free disk space during storing the data.

decimal separator: enter the character for separating floating point values within the input menus (ASCII-files always have the dot (“.”) as separator). The default character is given by the Windows settings. This character must be used if you enter a floating point value (e.g. 0,50 for "," or 0.5 for "."). The current setting is stored within the ini-file when leaving the program.

Bitmap size[kb]: this parameter controls the max.size of a bitmap created within REFLEXW. The default value is 16318 kb. The max. allowed bitmap size may vary from computer to computer. Therefore you may decrease this paramter if any problems occur. The program automatically restricts

the bitmap to this max. value and generates different bitmap parts if necessary.

BackgroundColor: choose the background color for the wiggle display.

Transp. comm.marker: if activated the comment marker textbox will be transparent

pick parameters: With this groupbox you may enter some parameters controlling the picking (see also chap. 6).
With the parameters **PickSymbols** you can choose between two different symbols for the picks (* and -).
The parameter **display every n.pick** controls that only every n. pick will be plotted. This does not affect the settings of the picks themselves. Enter a number larger than 1 if the phasefollower is too slow or if the picks are not displayed when scrolling fast.
With the parameter **pickcorrect traces** your may enter the number (one sided) of traces for the pick correction option corr.max/min.time (see also Pick MenuItem).
The parameter **corr.xy-coord bin(%)** controls the size of the crossing bin when your are using the option control xy-coord within the pick panel (see chap. 6). The parameter is in percentage of the current traceincrement. Increase this parameter if the option control xy-coord does not generate any picks.
With the option **show pickdifferences** activated the traveltime differences and thicknesses are displayed when using the difference pick.
With the option **keep distance** activated the x-position between the two picks are plotted in addition when using the difference pick. With this option not activated the x-position of the second pick is automatically moved to the x-position of the first one.
The option **decimal places** controls the number of decimal places for the ASCII-export of the picks.
The option **places** controls the number of places for the ASCII-export of the picks.
plot on 2.line: if activated the picks are also displayed on the 2.line whenever you press the plot or reset button. For example this option may serve as a controlling possibility of the picking if you have acquired the same data with two different frequencies.

core parameters: Within this groupbox you may enter some parameters controlling the coredata (see also option Core data/1D models under View MenuItem). The option **bar size** controls the size in pixels of the bars of the individual cores. With the option **show quality** activated a second bar is shown indicating a quality factor stored within the core datafile. With the option **show border** activated a black border rectangle is plotted in addition to the colored bar. With the option **show label** activated the labels of the cores are shown in addition. The label corresponds to the individual corename without CORE. With the option **save corefile** activated the current corefile is updated when storing the core velocity adaptations (see also velocity adaptation). With the option **save VLA-file** activated a VLA-ASCII velocity datafile is automatically created when storing the core velocity adaptations.

1.3 View MenuItem

ColorBars: activate this option for displaying the current color amplitude assignment bar at the right of the window.

Amplitudes: activate this option if you want the current amplitudes to be displayed in the status panel on the top (A:) when moving the mouse.

FileInfo: activate this option if you want the label infos about the loaded files shall be visible.

WiggleWindow: This option allows the additional plotting of the current trace or of the trace spectrum into a freely movable and scalable window. The y-axis corresponds to that of the profile. After activating the option you may choose between **current trace**, **curr. spectrum** or **2. line**.
After activating **current trace** the current trace is plotted in the wiggle-mode using the current Plotsettings. Activating the option **Tracenormalize** enables a normalization to the max. value of the current trace.
After activating the option **cur. spectrum** the frequency spectrum of the current trace is shown.
The current parameters time and amplitude are shown if the mouse cursor is inside the wiggle window.

Core data/1D models: activate this option if you want to display core data (1D-depth distributions) as vertical bars onto the line and onto the layershow depth profile. The core datafile should be stored under the path ASCII under the current project directory and must have the extension cor(e.g. c:\reflex\demodata\ascii\test.cor). A core datafile contains the informations of several cores associated with the current 2D-line. One core block within the file has the following format:

1. line: CORE + any identification
2. line: current location within the 2D-line (floating point format)
3.-x. line: layernumber thickness quality-factor mean-velocity layer-velocity

The dimension of the thickness and of the location is identical to the current distance dimension of the profile. The dimension of the velocity is m/ns or m/s respectively.
A block ends when the program finds the identifier CORE or at the end of the file.

example:

```
CORE   1 - dist.dim: METER, vel.dim: m/s
14.00
 1   7.887100  1    470.553    470.553
 2  11.804501  1    576.020    677.474
 3  15.052200  1    717.993   1059.677
CORE   2 - dist.dim: METER, vel.dim: m/s
24.00
 1   7.887100  1    470.553    470.553
 2  11.219000  1    573.388    677.474
CORE   3 - dist.dim: METER, vel.dim: m/s
34.00
 1   7.887100  1    453.631    453.631
 2  11.219000  1    562.827    677.474
 3  15.637701  1    713.371   1059.677
```

The max. number of different layers within one coreblock is 100 (identical to the max. number of layers within the layershow). The colors for the individual layerbars are identical to the colors within the LayerShow MenuItem. With the option **show quality** within the global settings activated a second

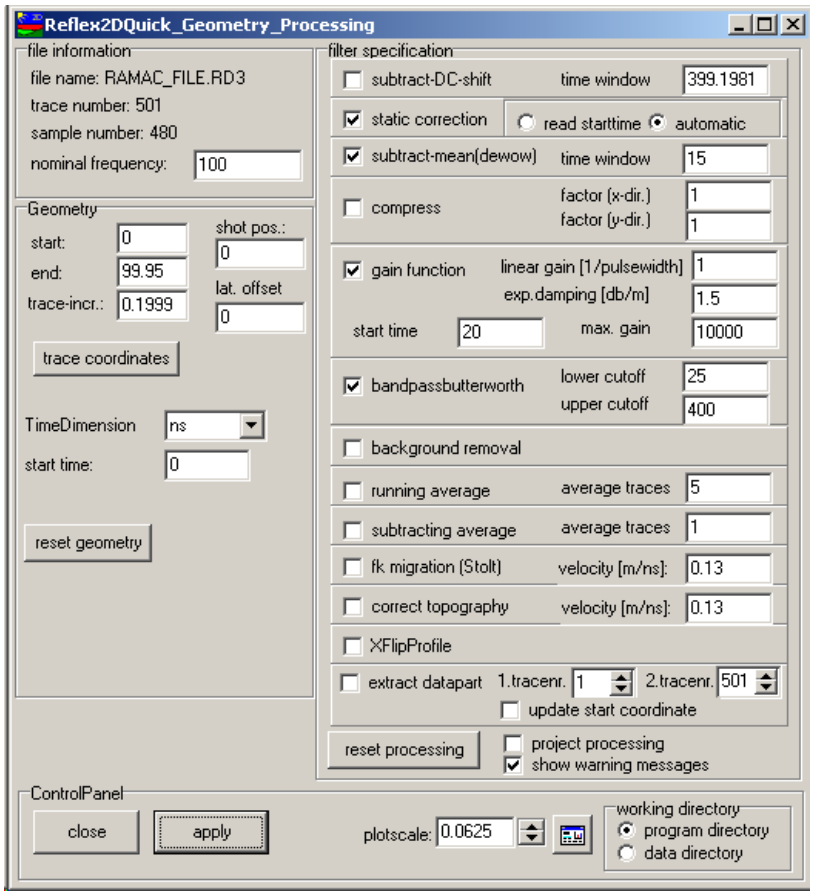
layerbar is shown indicating the quality factor with the following color assignment: 1 - white, 2-yellow, 3-red. Within the global settings menu you may also change the width of the layerbars and activate or deactivate some checking parameters.
These core data are also the basis for a coredata based velocity adaptation (see also velocity adaptation - suboption **core**).

profile line (trace header coord.): with this option activated the profile location based on the traceheader coordinates (see also trace header Edit, chap. 2.2) is shown in an additional window (any curvature of the line coordinates is displayed). When moving the mouse cursor within the data window the current xy-position of the mouse cursor is also shown.

TraceHeader axis: if activated the xy-receiver traceheadercoordinates are displayed along the distance axis in addition. The option is only available with deactivated plotoptions Rotate90degree and FlipXAxis. The number font is used. The number of places and the number of decimal places may be entered within the global settings menu within the pick parameters panel. The option transp.comm.marker within the global settings menu controls if the text will be transparent or if a white box will be underlied.
The suboptions at the top, at the bottom and at min.distance define the positions of the traceheader coordinates. With at the top activated the traceheader coordinates are displayed at the distance labels within the upper range of the profile below the distance axis using a 90 degree rotated font. With at the bottom activated the coordinates are displayed within the lower range of the profile again using a 90 degree rotated font. With at min.distance activated only the traceheader coordinate for the min. coordinate will be displayed above this min. coordinate. If enough space above the min. coordiante is available the x- and y-traceheader coordinates will be separately plotted, otherwise they will be combined separated by the "/" character (according to the options at the top and at the bottom).

2. Geometry and processing menu

After having loaded an **original datafile** the Reflex2DQuick_ Geometry_Processing menu opens. The filename, the number of traces, the number of samples and the nominal frequency are displayed. Here you may change the geometry and define a processing flow. The geometry/processing menu can also be entered at any later stage using the option or file/edit FileHeader.



The geometry is defined by the parameters **start**, **end** and **trace-incr**. The distance range (end - start) is always connected to the trace-incr. (traceincr. = distance / (number of traces-1)). Therefore when changing the parameters trace-incr. or start the end parameter is changed automatically, whereas when changing the parameter end the trace-incr. will be changed.
The **shot pos.** defines the position of the shot for single shot data (e.g. seismic shot for a refraction survey). For all other data (e.g. GPR common offset data) the parameter has no meaning
The **lat. offset** defines the lateral position of the 2D-line. It will only be used when exporting the picks using the ASCII-colums format..
The option **trace coordinates** opens a tabella containing the traceheader coordinates (see also chap. 2.2.1). The traceheader coordinates are necessary for storing the picks using any of the ASCII-formats (see also chap. 6.2).
The **timedimension** is either ns, µs or ms. By default the timedimension is set to ns for the following formats: Ramac, Radan, PulseEkko, Utsi and IDS-files. For the formats SEGY and SEG2 the timedimension ms is assumed. It is possible to change the default timedimension. The timeincrement and therefore the timerange does not change when changing the default timedimension except for:
- SEG2-format
- SEGY-format when changing to µs. When changing to ns it is assumed that the original timeincrement within the SEGY-format is given in ps (instead of the default µs) and therefore no changes are necessary.

The **start time** defines the time for sample 1 in the given time dimension Enter a negative value if the first arrival starts at later times..
The **distance dimension** is always meter.

The **working directory** can either be the program directory, this means where the program Reflex2DQick has been installed, or the current data directory, this means where the original data have been stored. The setting of the working directory will be kept if the program will be started next time.

Some **processing steps** (see chap. 2.1) are available within the filter specification panel. By default no

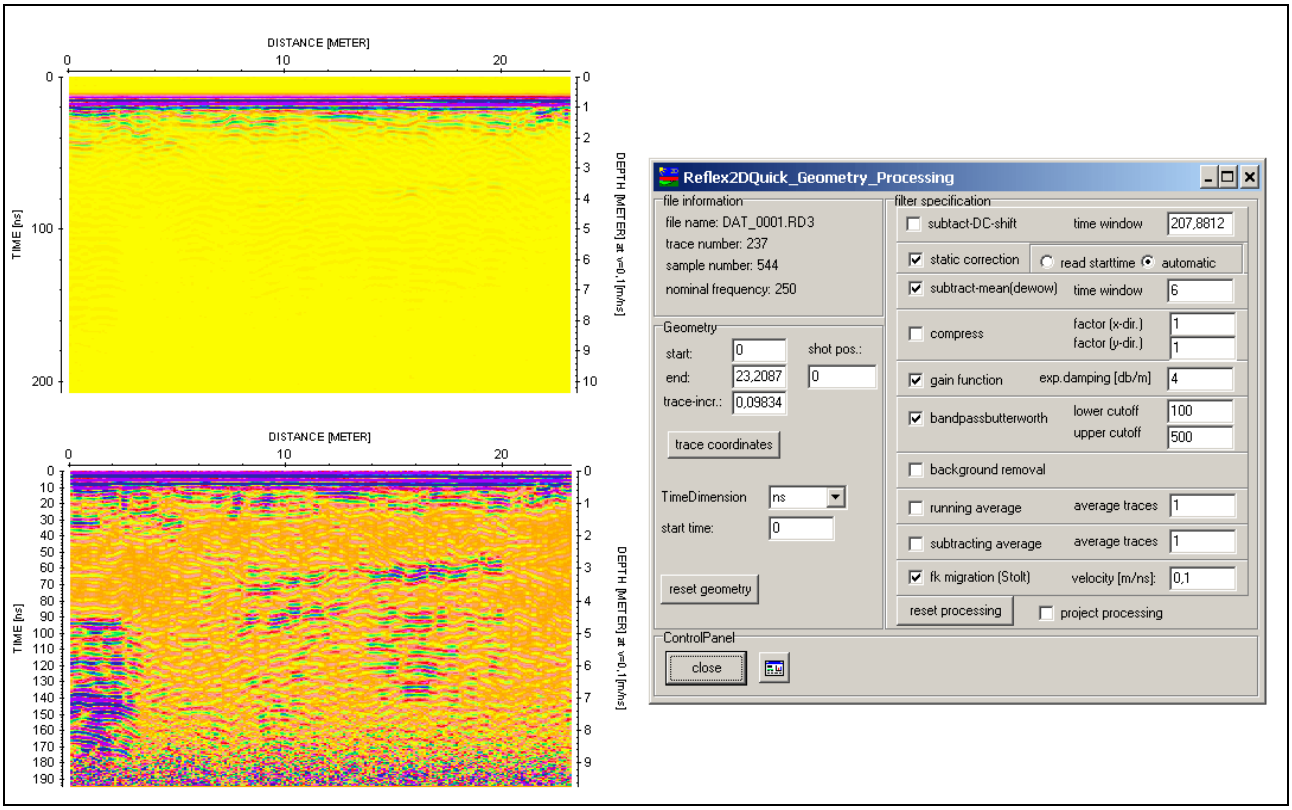
processing step is active and the filter parameters have been set to default values. The default values are automatically calculated from the currently loaded file. The option **reset processing** deactivates all chosen processing steps and sets the filter parameters to the default ones. The option  enters the PlotOptions menu (chap. 3).

The program automatically controls whether a processing/geometry file for the chosen original file is present. If the file is present a query appears whether the informations from this file shall be loaded or not. You should not load these informations if you used the same filenames and working directory for an older project. The query can be deactivated using the option **show query if geometry file exists**. Such a processing/geometry file is stored under the path procdata under the Reflexw 2D-Quick **working directory**. It has the same name like the original datafilename (without path and without extension) and the extension par (Test.dzt.par e.g.). Note: These parameter files are not deleted automatically closing Reflexw2DQuick. Please delete them manually to recover disk space. If such a parameter file exists the geometry and the processing flow as well as the plotoptions are read in from this file and the parameters are displayed within the Reflexw2DQuickGeometry_Processing menu.
message dlg if geometry file exists if activated a message dialog appears if the processing/geometry parameterfile for the loaded datafile already exists
It is also possible to read the processing flow from an external project processing file. For that purpose you have to activate the option **project processing**. Then the wanted project processing file is queried.
With the option **show warning messages** deactivated warning messages for some processing steps will not displayed.
After having activated the option **close** the processing flow will be applied on the current file. This may take some processing time but each subsequent replotting does not need the application of the processing flow whereby a fast display is given.
Using the option **apply** the processing flow will be applied on the current file without closing the Geometry_Processing menu. This option might be useful for testing the processing flow.
The processing/geometry file is automatically updated. With the option project processing activated the project processing file is updated.
Note: To compare the processed data with the original data you may open Reflex2DQuick a second time and load the original data set.
If you change the processing flow and apply it to the data, again the original data set (and not the currently displayed processed data set!) is processed.

The **plotscale** option allows to enter a multiplication factor for the color-amplitude assignment or for the wiggle size. The minimum and maximum amplitude values are controlled by the multiplication factor Amplitudescale. With a value of 1 for Amplitudescale the amplitudes range from -2048 to 2048 for unnormalized data and from -1 to 1 for tracenormalized data. With a Value of 0.0625 the amplitudes range from -32768 to 32768 (see also Amplitudescale under Pointmodeattributes).

To print the files see chap. 4.

To export/save the original data set or the processed data set see chap. 7.



Comparison of the original raw data (upper panel) with the processed data (lower panel) based on the settings given in the right geometry_processing menu.

2.1 processing steps

The following processing steps are available:

subtract DC-shift: With this option activated a so-called zero mean, i.e. the subtraction of an existing time constant shift is calculated for each trace. As filter parameter the timewindow must be entered. Within this time range (aloways starting at the start time) the mean is calculated for each trace which is subsequently subtracted from all samples of each trace. Therefore it has to be guaranteed that the mean value in the corresponding time range complies with the shift you want to eliminate.
Seismic shot data often reveal such a DC-shift which will be best eliminated using this filter.

static correction: With this option activated a static, this means a time-independent correction for each trace in time direction will be done. The time shift will either be read from the entered start time (option read starttime activated) or will be automatically determined from the data (option automatic activated). The data above the time shift level will be lost.
The active suboption automatic offers the possibility to automatically determine the first significant arrival (independent from the polarity) and to correct the start position of the time axis to this phase. The internal input parameters are fixed to a threshold of 10 % of the max. existing amplitude value and to the dominant frequency of the wavelet. If the polarity of the automatically determined onsets has been changed for one or more traces a warning message appears. An automatic resampling in time direction for each trace is included during the processing in order to increase the resolution of the automatically determined phase. The identified time is shifted to smaller times to the zero crossing.
Use this option for example for a fast static correction of the first arrival (e.g. the direct wave from source to the receiver) if this phase shows some irregularities (e.g. a drift or trigger fluctuations) or to automatically change the start time if there is a lag between start of scan and first arrival.

subtract mean (dewow): With this option activated a running mean value is calculated for each value of each trace. This running mean is subtracted from the central point. As filter parameter the time window for the calculation of the running mean value must be entered. When first loading the wanted dataset the program automatically determines a default value for the time window. The window range should be set to about one principal period.
This filter may be used for eliminating a possible low frequency part (dewow) which may especially occur for GPR data.

compress: This option allows both a compression in time-(Y-)direction and in distance-(X-)direction. The filter parameters factor (x-dir.) and factor (y-dir.) define the size of the compressing window in x- and y-direction, this means every which trace and every which sample will be used.
A compression might be reasonable before the application of computing-time intensive processing steps (e.g. fk-migration). Of course the aliasing criterium must be considered.

gain function: The filter facilitates the possibility of multiplying the data points by a given function g(y) or g(t) respectively. The function g(t) consists of a linear and an exponential part: $g(t)=(1+a*t)*e^{(b*t)}$ with a=a'/pulse width and b=b'*v/8.69 with v=0.1 m/nsec or 1.0 m/msec respectively. The pulse width is automatically taken from the nominal frequency if given (see option FileHeader Edit). Otherwise, the nominal frequency is automatically determined from the first arrival. The two filter parameters a'(linear gain) and b'(exp. damping) must be entered. a': not dimension except for the case of non set nominal frequency; b': input in dB/meter. Default values are 1 for the linear gain and 5 for exp. damping. In addition you have to enter the start time (the filter starts at that time with value 1) and the max. gain.
The data are multiplied by this function in order to compensate for possible damping or geometric spreading losses.

bandpass butterworth: Here you can apply a bandpass filtering in the time domain using a recursive filter. The filter band is specified by the setting of two frequency values. The first point determines the lower cutoff frequency, the second one the higher cutoff frequency. The frequency spectrum below the low cut and above the high cut frequency is set to zero. By the corresponding choice of the points of the bandpass either a lowpass or a highpass can be approximately realized.
Noise can be suppressed with the bandpass filter when it differs from the signal in its frequency content.

background removal: This filter performs a subtracting of an averaged trace, a so called background removal. With this option you can eliminate temporally consistent noise from the whole profile and therefore possibly make signals visible, previously covered by this noise. This filter method also suppresses horizontally coherent energy. Its effect is also to emphasize signals which vary laterally (e.g. diffractions). Attention: it might happen that the filter causes non real signals. This holds true when the averaged time series contains energy which is not present within any part(s) of the profile.

running average: The filter performs a running average over a choosable number of traces for each time step. The running average is performed over a number of traces (parameter average traces). The max. bandwidth is restricted to 256 traces to be averaged. For a bandwidth of 4 the current sample, the next two in horizontal direction to the left and the next two in horizontal direction to the right, i.e. five samples for each time value, are taken into account. The filter is the more effective the larger the selected bandwidth.
This filter method suppresses trace dependent noise. Its effect is to emphasize horizontally coherent energy.

subtracting average: The filter performs a subtracting average over a choosable number of traces for each time step. The filter performs a so called sliding background removal. The subtracting average is performed over a number of traces (parameter average traces). The max. bandwidth is restricted to 256 traces to be averaged. For a bandwidth of 4 the current sample, the next two in horizontal direction to the left and the next two in horizontal direction to the right, i.e. five samples for each time value, are taken into account. From these five samples the mean value is calculated. This mean value is subtracted from the value of the current sample and the result is assigned to the current sample as new value. The filter is the more effective the smaller the selected bandwidth.
This filter method suppresses horizontally coherent energy. Its effect is to emphasize signals which vary laterally (e.g. diffractions).

fk migration (Stolt): A 2D fast fk-migration on the basis of a constant velocity is performed. The method works in the frequency-wavenumber (fk) range. Within the fk-range a variable transform is done based on the entered constant velocity (frequency is transformed onto the vertical wavenumber). First the x-t data are tranformed into the f-k-range. After having done the transformation the migration process will be done and after that the back transformation into the x-t-range is performed.
The profile must represent a so called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source". A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions are present the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

Topographic correction: A static, this means a time-independent, correction for each trace in time direction based on an ASCII-file is performed.
Each line of the ASCII-file with the topographic coordinates contains one set of 3D coordinates:
x-position y-position z-position
The y-position must alsways be 0. The x-position defines the location point on the profile line. The z-

position defines the topographic niveau. At least one blank must exist between the individual coordinates. Example with 3 different coordinate sets:
0 0 10
1 0 9
2 0 9.5
The dimension of the coordinates must be the meter.
The filter parameter velocity is used to transform the z-values of the topographic coordinates into a two-way travel time. The coordinates are taken from the given geometry settings (start, end and trace-incr.). A spatial linear interpolation is done between the given topographic points.
The program automatically determines the individual topographic values from the given topography matrix. For this purpose the next topographic coordinate set within each quadrant is determined for each trace location. Afterwards a weighted mean of the single z-positions is determined which is used for the static correction for this trace.
The greatest z-position (transformed into two-way travel time) is taken as the base level. The trace with this correction value is not modified in time. All traces are shifted down (i.e. towards larger times) relative to this base level according to the difference of its correction values and the base level. The inserted time range is filled with zeros. This means that the area between the current topographic niveau and the maximum niveau is filled up with material with the given velocity.

XFlipProfile: The option gives the possibility to flip a profile in x-direction. The option might be useful if the 2D profiles have been acquired using a meandering scheme. Then for example every 2. profile must be flipped in x-direction in order to get the same profile direction for all profiles. The entered start coordinate will not be changed.

extract datapart: allows to extract a special datapart. For that purpose you must enter the 1. and the last tracenumber of the datarange to be extracted. The option might be useful if the data at the beginning and/or at the end are of no importance (e.g. a GPR profile acquired without a wheel). The wanted tracenumbers can be taken from the panel at the bottom of the profile (view option amplitudes must be activated). The option can also be used for extracting a datapart from GPS guided data. For that purpose you may activate the option view/profile line and you may define the wanted tracenumbers when moving the mouse within the profile. The actual x/y position is shown within the schow line position window.
With the option update start coordinate activated the start coordinate of the profile will be updated depending on the chosen first trace and the entered traceincrement. With deactivated option the start coordinate will not be changed.

2.2 Traceheader Edit

Within this menu you may show and update the trace header informations of each trace of the current profile.

trace number: specifies the trace number within the profile to be edited.

distance: shows the distance along the profile

shot x-pos: shows the shot position in x-direction

shot y-pos: shows the shot position in y-direction

shot z-pos: shows the shot position in z-direction (elevation)

rec. x-pos: shows the receiver position in x-direction

rec. y-pos: shows the receiver position in y-direction

rec. z-pos: shows the receiver position in z-direction (elevation)

CMP x-pos: shows the CMP position in x-direction

CMP y-pos: shows the CMP position in y-direction

ensemble-nr.: this number is used for subdividing the profile into different 2D-lineparts, e.g. a 3D-file consisting of different parallel 2D-lines. All traces belonging to one 2D-line have the same ensemble-nr. (see also option Create(3D)Ensembles).

field record nr. (component): shows the field record nr. of the original data.

Timecollect: shows the original time when recording the trace

SaveOnASCII: the option allows to save the traceheader coordinates on an ASCII-file. Activating this option opens a new window where you may define which parameters shall be written on the ASCII-file. Each line of the ASCII-file contains the desired informations of the individual trace. In addition to the parameters mentioned above two additional parameters can be written out: marker and TimeCollect. Marker values > 0 and < 100 define a distance marker, marker values >= 100 a comment marker. TimeCollect is the acquisition time in seconds after midnight. This value may be used in order to synchronize time based GPS-data (see option **update** based on **GPS-times**).

update: The group box allows to update the geophone-, shot- and CMP-coordinates based on the distance coordinates of the file header (option **type** is set to **fileheader** - coordinates in profile direction and in profile constant(for a 3D-datafile you should use **fileheader 3D**) or based on a special circle geometry (option **type** is set to **circle**) or the coordinates are read from an ASCII file (option **type** is set to **ASCII-file**).

The options **interpolation** and **interpolation 0-data** and **interpolation equal data** allow the interpolation of the position data stored in the trace header, if for various traces no corresponding values exist.
The option **interpolation** uses a special flag within the traceheader of each trace which druing the import of the data is set to 0 by default and set to a special value when traceheadercoordinates have

been recognized. Only these traces where this flag is 0 are interpolated.
The option **interpolation 0-data** controls if all traceheadercoordinates of one trace are 0 and performs an interpolation for those traces.
The option **interpolation equal data** controls if identical coordinates for the x- and y- shot and receiver positions for subsequent traces are present and performs an interpolation for those traces.

If the type **circle** is activated, you must enter the **radius** r and the x- and y-midpoint coordinates (**x-mid** and **y-mid**) of the circle. The circle is subdivided into equidistant angle ranges. Based on these values the x- and y-CMP coordinates are calculated using the following formulas: $x=r*\sin(alpha)+x_{mid}$ and $y=r*\text{sing}(alpha)+y_{mid}$ with alpha ranging from 0 to 360 degrees.

With the type **ASCII-file** activated you must choose the wanted DST file from the file list opened when entering the option update. Any ASCII file must exist under the path ASCII and must have the extension DST. Each line of the ASCII file contains the following 6 or 8 (z-pos. optional) informations:

trace number distance Shot-X-Pos Shot-Y receiver-X receiver-Y shot-Z(opt.) receiver-Z(opt.)

Defining the trace number gives you the opportunity to read the geometry only for a distinct part of the data. If the distances within the ASCII-file are zero (2.colum) they will be calculated from the total difference between each shot and receiver positions (x,y and z-coordinate are taken into account).

With the type **ASCII-file/interpol.** activated the data are read again from an ASCII-file (see type ASCII-file). In addition an automatic interpolation is done if for various traces no coordinate data are present within the ASCII-file.

With the type **RAMAC-borehole** activated it is possible to read the source and receivers coordinates from the ASCII RAMAC-borehole files. First you must choose the wanted tlf file. The corresponding f1.fot and f2.fot must exit.

With the type **RAMAC-GPS** activated it is possible to read the GPS coordinates using the format of the RAMAC GPS device. The file should have the extension COR. Each line of the GPS-ASCII file contains the following informations (following optional additional informations are ignored):
tracenumber,date,time,Northing,West,altitude,PDOP
The separator between the different values may be either comma, blank or tab. In addition the orientation may follow the coordinate value seaparated by a comma, blank, tab or nothing (e.g.53.5891830N or 53.5891830,N).
The latitude coordinates are set to negative values for Northing equal ‘S’ and positive input values or Northing equal ‘N’ and negative input values. The longitude coordinates are set to negative values for West equal ‘W’ and positive input values or West equal ‘E’ and negative input values.
REFLEXW stores the northing coordinates, the west coordinates and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the *.cor file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers).

With the type **WSKTRANS-GPS** activated it is possible to read the GPS coordinates using the format of the WSKTRANS software. The file should have the extension ut or utm. The file consists of two optional header lines and then each line of the GPS-ASCII file contains the following informations (following optional additional informations are ignored):
tracenumber Northing West altitude
The separator between the different values may be either comma, blank or tab. REFLEXW stores the

northing coordinates, the west coordinates and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the *.ut(m) file these coordinates are automatically be replaced with an interpolation.An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers).

With the type **CSV-GPS** activated it is possible to read the GPS coordinates using the format of the CSV-GPS software. The file should have the extension csv.The file consists of one header line and then each line of the GPS-ASCII file contains the following informations:
latitude,longitude,altitude,tracenumber
REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the *.csv file these coordinates are automatically be replaced with an interpolation.An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers).

With the type **IDS-GPS** activated it is possible to read the GPS coordinates using the format of the IDS-GPS software. A GPS ASCII-file with the extension gps containing the GPS-coordinates must exist. The corresponding tracenumbers are taken from the comment markers with the comment "gps". Normally the original IDS-file contains these comment markers and they are taken over into the Reflexw file during import. Older IDS-systems generate an additional ASCII-file with the extension gpt containing the tracenumbers. If this file exists (same filename like the gps-file except the extension) the tracenumbers are taken from this ASCII-file.
There might be a header within the gps-file but the first line containing the first GPS-coordinates corresponds to the first tracenumber stored within comment marker or within the first line of the gpt-file. The next line corresponds to the second tracenumber and so on.
One line of the gps-file contains the following informations:
*$GPGGA,123015.00,5141.607563,N,00439.680658,E,2,04,2.3,2.74,M,47.21,M,2.8,0000*40*
whereby the following fields will be interpreted:
field 3: 5141.607563 - geographic latitude in *ddmm.mmmmmmm* format (51 degrees and 41.607563 minutes)
field 5: 00439.680658 - geographic longitude in *dddmm.mmmmmm* format (4 degrees and 39.680658 minutes)
field 10: 2.74 - antenna height above MSL (mean sea level) reference (2.74 m)
The latitude and the longitude are stored in degress within the x- and y-receiver traceheader positions.
REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined.

With the type **PulseEkko-GPS** activated it is possible to read the GPS coordinates using the format of the PulseEkko-GPS software. One GPS file with the extension gps must exist. Each block of the file starts with a line containing the trace number. A number of lines follow containg the gps-coordinates. By default the coordinates are taken from the line starting with $GPGGA. Following an example of one GPS-block is shown:
Trace #1 at position 0.000000
*$GPVTG,266.2,T,,,007.12,N,013.19,K,A*4E*
*$GPGSA,M,3,03,19,15,16,18,22,,,,,,,2.8,1.7,2.2*33*
*$GPRMC,093704,V,4625.938530,N,00950.727816,E,007.12,266.2,240404,0.7,W,N*2D*
*$GPGGA,093705.00,4625.938360,N,00950.724940,E,1,06,1.7,2635.92,M,48.00,M,,*53*
Within the $GPGGA line the following fields will be interpreted:

field 3: 4625.938360 - geographic latitude in *ddmm.mmmmmm* format (46 degrees and 25.938360 minutes)
field 5: 000950.727816 - geographic longitude in *dddmm.mmmmmm* format (9 degrees and 50.727816 minutes)
field 10: 2635.9 - antenna height above MSL (mean sea level) reference (2635.92 m)
The latitude and the longitude are stored in degress within the x- and y-receiver traceheader positions. REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined.

With the option **tabella** activated the current traceheader coordinates are shown within a tabella and can be changed manually. The option save changes saves the current settings of the traceheader coordinates. The option reload from file reloads the traceheader coordinates from file.The option save on AsciiFile allows to write the coordinates on an ASCII-file with the extension DST (see option ASCII-file above).

The option **GPS-times** allows to update the traceheader coordinates based on the acquisition times (in secs after midnight) stored within the line traceheader and an ASCII-file containing the GPS-coordinates and the GPS-time. The format of the ASCII-file is as following:

Easting	Northing	MSL	GPS Date	GPS Time
638434,601	7215215236	143.227	19.05.02	07:32:12
638434.573	7215215253	143.191	19.05.02	0,31403935185

There is no header line - the ASCII-file must begin with the first data values. The GPS-date must be present but it will not be used for the matching. Only the GPS-times are used. The program performs an automatic interpolation or extrapolation if the times are not exactly matching.

The option **calculate distances** allows to update the traceheader distances based on the x- and y-receiver traceheader coordinates. The start distance is taken from the fileheader but can be changed manually (option start distance).

When using the option **exchange rec. -> distance** the distance traceheadercoordinate is set to the x-receiver coordinate for each trace.

Using the option **subtract constant value** allows to subtract a constant value (x and y independently) from the x- and y-shot and receiver positions.

The option **UTM-conversion** allows to convert GPS degree traceheader coordinates to local cartesian coordinates using the so called Universal-Transverse-Mercator(UTM)-conversion (WGS84). You must specify where the latitude is stored within the traceheader (latitude in degrees either in x- (shot x, rec x and CMP x-pos) or y-coord). The longitude is automatically vice versa in y- or x-coord.
The conversion can be done automatically (default) or by using a distinct zone. If the automatic conversion is used, the UTM-zone for the whole profile is determined using the GPS traceheader coordinates of the first trace. So the coordinate origin is fixed for the whole profile not regarding, that the profile may lie in different UTM-zones.
If a distinct zone shall be used for the conversion, it can be chosen from a list if the option 'distinct zone' is chosen. This zone is also fixed for the whole profile.
There exist 120 different zones which cover nearly the whole globe: 60 zones cover the northern part of the world (0 deg <= lat <= 84 deg), the rest the southern part (-80 deg <= lat < 0 deg).
The zones of every half sphere are determined as following:

Zone 1: -180 deg <= lon <= -174 deg (if lat >= 0 this is zone utm1n, otherwise utm1s)
Zone 2: -174 deg < lon <= -168 deg
..
Zone 60: 174 deg < lon <= 180 deg (if lat < 0 this is zone utm60s, otherwise utm60n)

For the regions near the poles (Lat > 84deg or Lat < -80deg) the Universal-Polar-Stereographic(UPS)-conversion is performed instead the UTM-conversion.

General:

The GPS-coordinates are stored within the x- and y-receiver traceheader positions (rec-x and rec-y). In the case of the datatype "constant offset" the shot and the CMP-coordinates are set equal to the receiver positions. In the case of the datatype "single shot" the shot coordinates are set to the entered shot position and shot offset defined within the fileheader. The CMP-coordinates are always calculated from the receiver and shot positions.

The single coordinates stored in the individual traceheaders are necessary for some processing steps, e.g. for Export into the format ASCII-3COLUMS, the ascii-columns output of the picks with the option xy-coordinates activated (see also Pick save MenuItem).

The option **calculate distances** allows to update the traceheader distances based on the x- and y-receiver traceheader coordinates. The start distance is taken from the fileheader but can be changed manually (option start distance).

When using the option **exchange rec. -> distance** the distance traceheadercoordinate is set to the x-receiver coordinate for each trace.

Using the option **subtract constant value** allows to subtract a constant value (x and y independently) from the x- and y-shot and receiver positions.

project y to xz-plane: allows to project the y-coordinates of the individual shots and receivers onto the xz-plane. The distance between source and receiver remains unchanged. Therefore the x- and z-coordinates are changed respectively.

project z to xy-plane: allows to project the z-coordinates of the individual shots and receivers onto the xy-plane. The distance between source and receiver remains unchanged. Therefore the x- and y-coordinates are changed respectively.

Within the groupbox **Create(3D)Ensembles** it is possible to subdivide the current profile into different lineparts. The ensemble-nr. within the traceheaders controls the size of each linepart.
With the option **equidistant** activated the current profile will be subdivided into different equidistant lineparts. The option **nr. of traces** defines the number of traces per linepart (ensemble).

With the option **constr. change markers** activated the current profile will be subdivided into different lineparts at the given construction change markers positions.
Activate the option **create** to generate the lineparts (the Ensemble-Nr. within the traceheaders will be updated). This option allows you to redefine the individual ensembles (2D-lines) of a 3D-file or of a 2D-file consisting of different parts.

close: terminates this window

2.2.1 traceheader tabella

A tabella containing the traceheader coordinates and the traceheader gain opens either when using the option update together with the updatetype tabella within the traceheader menu or after having activated the option trace coordinates within the Geometry_Processing menu.
The tabella shows the traceheader coordinates (distance and the x, y, z-coordinates of the receivers and the shots) and the gain. These values can be manually changed. To be considered for manual editing: distance value and rec.-x value should be equal for each trace.

The option **save changes** saves the manual changes within the traceheaders of the current file.

The option **reload from file** reloads the coordinates from the traceheaders of the current file. Possible manual changes will be lost.

The option **update from fileheader** allows to update the receiver and shot-coordinates based on the current fileheader coordinates.

The option **update fileheader** allows to update the fileheader coordinates based on the traceheader coordinates. The option is useful in order to fit the fileheader and the traceheader coordinates.

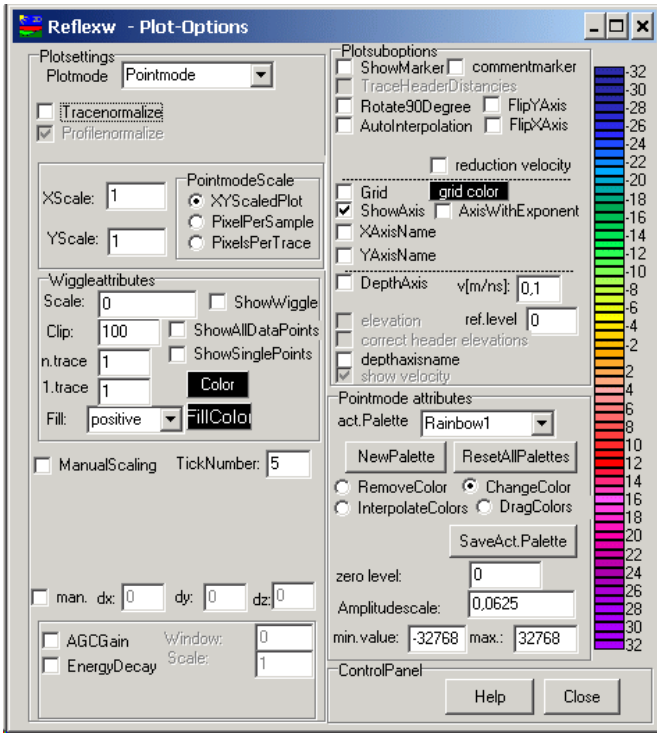
The option **save on AsciiFile** stores the traceheader coordinates on an ASCII-file with the extension dst. Each line of the ASCII file contains the following 8 informations:
tracenumber distance Shot-X-Pos Shot-Y-Pos receiver-X-Pos receiver-Y-Pos Shot Z receiver Z

The option **load from AsciiFile** allows to load the traceheader coordinates from an ASCII-file with the extension dst. Each line of the ASCII file contains the following 8 informations:
tracenumber distance Shot-X-Pos Shot-Y-Pos receiver-X-Pos receiver-Y-Pos Shot Z receiver Z
An automatic interpolation is done if for various traces no coordinate data are present within the ASCII-file.

The option **interpolate** allows to interpolate between two given coordinates. For that purpose activate the option interpolate, then click on the wanted first coordinate and then on the second one. Inbetween a linear interpolation will be done. The interpolation is only possible for one single traceheader coordinate group (for example only rec-z).

3. PlotOptions

Within this menu you may change the current plot settings.



3.1 Plotsettings

Plotmode: The option allows the specification of the plottype. You may choose between pointmode and wigglemode.
Pointmode means that the data are plotted with colored pixels according to the predefined color-amplitude configuration (see also pointmodeattributes).
Wigglemode means that each trace of the profile is plotted as a polygonal line. The size of the deflections of each wiggle is controlled by the parameter scale (see also Wiggleattributes). The two modes might be used together if the pointmode is chosen together with the activated option **ShowWiggle**.

Three different scale modes are incorporated:

XYScaledPlot: Activating this option means that the data are completely plotted into the current window provided that the two scale options **XSCALE** and **YSCALE** are set to 1. Increasing the number of the scale options means a zooming up, decreasing means a zooming down.

PixelsPerSample: Activating this option means that the plotting size of each data point is given in screen pixels. The size in x-(distance) direction and y-direction can be changed using the option **XSCALE** and **YSCALE** respectively.

PixelsPerTrace: Activating this option means that the distance between successive traces is given in screen pixels (option **XSCALE)**. In every case the complete time series of each trace is plotted corresponding to the size of the current window. This means no zooming possibilities in y-(normally time-)direction are given for that scale mode.

XScale: enter a value for the window scaling in x-(normally distance-)direction. If PixelsPerTrace or PixelPerSample is activated, the parameter gives the distance between successive traces in pixels. If XYScaledPlot is activated, the parameter gives the x-zooming value corresponding to the current profile window (e.g. a value of 2 means that half of the traces are plotted into the current window).

YScale: enter a value for the window scaling in y-(normally time-)direction. If PixelPerSample is activated, the parameter gives the range between successive points in pixels. If PixelsPerTrace is activated, the parameter has no meaning. If XYScaledPlot is activated, the parameter gives the y-zooming value corresponding to the current profile window (e.g. a value of 2 means that half of each trace length is plotted into the current window).

Tracenormalize: Activate this option if you wish the data to be plotted amplitude normalized for each trace. With the option profilenormalize deactivated (see below) the maximum amplitude of each visible trace is normalized to 1. Like that a plotting is guaranteed, where all traces are well visible.

Deactivate this option if you want to plot the data with real amplitudes. For the **wigglemode** the size of the individual deflections is controlled by the parameter Scale (tracenormalize: Scale = Size in Pixels, no tracenormalize: current amplitude * Scale = Size in Pixels). For the **pointmode** the parameter Amplitude scale controls the amplitude color assignment.

Profilenormalize: with this option activated the normalization is not done based on the max. amplitudevalue of each trace but on the mean amplitudevalue of the complete profile. Thereby amplitude variations from trace to trace within one profile will remain but it is possible to compare profiles with different value scales. The option is only available with the option tracenormalize activated. To be considered for
Wigglemode: The entered wiggle size corresponds to the mean amplitude. Therefore you must enter a clip value greater than 100 for the greater amplitudes. Otherwise these amplitudes will be clipped. By default a clip value of 200 is set.
Pointmode: by default the amplitudescale is set to 0.5 - this means that the color amplitude assignment includes all amplitude values until twice the mean amplitude. Higher amplitude values are assigned to the max. color(s).

Manual scaling: Activate this option if you wish to manually input the min. and max. axis lengths in x- and y- (time-) direction (see also chap. 3.4). Deactivate this option if you wish the program to automatically determine the axis lengths.

Man.: if activated the manual subdivision of the axis (options dx, dy and dz, see also chap. chap. 3.4 - manual scaling and incrementation) will be enabled - if deactivated the program automatically chooses a suitable incrementation.

TickNumber: Enter the number of ticks between the axis marks.

3.2 Pointmodeattributes

This group box controls the attributes for the pointmode. The color amplitude assignment consists of 128 different colors which are linearly distributed between a minimum and maximum amplitude value. The minimum and maximum amplitude values are controlled by the multiplication factor **Amplitude scale**. With a value of 1 for Amplitude scale the amplitudes range from -2048 to 2048 for unnormalized data and from -1 to 1 for tracenormalized data. With a Value of 0.0625 the amplitudes range from -32768 to 32768.

act.palette: load the wanted color palette from the stored palettes. The following palettes are predefined: Rainbow1, Rainbow2, Gray1, Gray2, Gray3, BlueGrayRed. You may add any changed or created palette by activating the option **SaveActPalette**.

NewPalette: enter this option for defining a new palette. Please choose 16 different colors from the color dialogue menu. Between these 16 colors a linear interpolation is automatically done in order to define the 128 different colors. The new color palette is shown in the color bar on the right-hand side.

ResetAllPalettes: this option resets all palettes to the default ones.

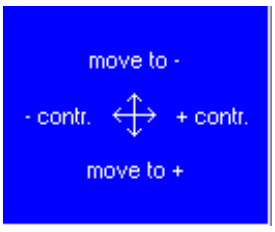
SaveAct.Palette: this option allows to save the current palette on disk with a freely choosable name.

RemoveColor: with this option activated you may remove any color from the color bar on the right-hand side by clicking on the corresponding color using the left mouse key.

ChangeColor: with this option activated you may change any color from the color bar on the right-hand side by clicking on the corresponding color using the left mouse key. The new color must be defined in the opened color dialogue menu.

InterpolateColor: with this option activated you may interpolate the colors between two choosable colors. Click on the two colors in the color bar using the left mouse key. After having chosen the second color, the in-between colors are linearly interpolated.

DragColor: two possibilities to interactively drag the colors are given



- a blue panel opens (see picture on the right) which allows to interactively change the colors when moving the mouse with pressed left mouse key within the panel. Starting from the midpoint increases the contrast to the right and decreases it to the left. The color table is moved to the colors which are assigned to negative amplitude values when moving to the top and vice versa when moving to the bottom. The changes are instantaneously shown within the loaded profile.
- you may continuously move a linearly interpolated color range. After having activated this option, please click on two different colors. In-between these colors a linear interpolation is done. The trackbar below may be used in order to increase or to decrease this color range.

The wanted color palette must be saved for a later use. Otherwise the original palette will be reloaded whenever you are entering the profile.

Zero level: enter a zero amplitude level for the color-amplitude assignment. The mean level of the color amplitude assignment is shifted by this given level. Thereby the color amplitude assignment is defined by this parameters and the value given for the Amplitude scale. A value 0 for zero level is useful for “normal” seismic or GPR-data. A value different from 0 is useful e.g. for displaying data with only positive or negative amplitude values (e.g. timeslices calculated on the base of absolute or envelope data).

Amplitude scale: The minimum and maximum amplitude values are controlled by the multiplication factor Amplitude scale. With a value of 1 for Amplitude scale the amplitudes range from -2048 to 2048 for unnormalized data and from -1 to 1 for tracenormalized data. With a Value of 0.0625 the amplitudes range from -32768 to 32768 provided the value of zero level is set to 0. The min. and max. Amplitude values are shown within the options **min. value** and **max. value** which also may be used for defining the amplitude scale. If new values will be entered here both the amplitude scale and the zero level will be updated accordingly.

autom.scale: the option allows to automatically define the min./max. amplitude range for the color/amplitude assignment. If activated the program extracts the min. and max. amplitudes from the actual profile or slice and uses these values for setting the color palette. The amplitude scale cannot be changed in this case any more.

auto 0-symmetry: If the option autom.scale is activated the 0-level of the color palette may be different from profile (slice) to profile (slice) as the min./max. values may vary. The option auto 0-symmetry overcomes this problem. If activated the absolute max. value will be determined from the min./max. values and either the min. or the max. value will be set to the absolute max. value with the corresponding sign. Activating this option might be useful if "normal" profiles are displayed with positive and negative amplitudes whereas deactivating this option might be a good choice for plotting slices showing the envelope.

3.3 Wiggleattributes

Scale: enter a value for the size of the individual wiggle deflections. The size is calculated by: current amplitude * Scale = Size in Pixels. If the option Tracenormalize is activated, the current amplitude must be replaced by the value 1 - therefore the size directly corresponds to the wigglesize in pixels of the max. or mean (profilenormalize activated) amplitude.

ShowWiggle: activate this option if you want to display the polygonal line.
With the pointmode selected this option controls if the data are plotted in pointmode together with the wiggles or not.
With the Wigglemode selected the option controls if the polygonal line is plotted or not. With this option deactivated only the filled part is plotted in this case (see below).

ShowAllDataPoints: activate this option if you want to display all data points of the trace even if the number of data points per trace is larger than the pixel number of the current window. Deactivating the option means that every trace is resampled based on the current screen points. If the number of samples to be displayed is much larger than the screen point number aliasing may occur. In this case the option should be activated. This means that every data point is plotted.

ShowSinglePoints: if activated the datapoints are marked by a cross symbol in addition.

n.trace: only every n.trace is plotted in wiggle-mode. This option is for example useful if you want to plot a large data file in point mode together with the wiggles.

1.trace: specifies the first trace within each block to be plotted if n.trace is greater than 1. The value of 1.trace must be between 1 and n.trace.

Color: choose the color for the polygonal line of the wiggles.

Fill: controls the filling of the polygonal lines.
Entering **no**, means no filling.
Entering **positive**, means only the positive amplitudes are filled with the chosen FillColor.
negative means only the negative amplitudes are filled with the chosen FillColor.
colors means that the wiggles (positive and negative amplitudes) are filled with the current color amplitude assignment. In this representation correlated signal arrivals are very well recognizable.
Pos./neg. means that both the positive and negative wiggles are filled using the current fill color.

Clip: With the parameter clip all amplitudes, exceeding the value of clip in pixels, can be cut. A large value for width and a small value for clip thus facilitates to recognize signal arrivals of small amplitudes otherwise covered by phases with large amplitudes.

FillColor: choose the color for the filled areas of the polygonal lines. The Fill option must be set to positive or negative.

3.4 Manual scaling and incrementation

xmin: enter a value for the minimal x-(normally distance-) value of the line to be plotted

ymin: enter a value for the minimal y-(normally time-) value of the line to be plotted
enter a value for the maximal axis value in x-direction

xmax: enter a value for the maximal x-(normally distance-) value of the line to be plotted

ymax: enter a value for the maximal y-(normally time-) value of the line to be plotted
enter a value for the maximal axis value in x-direction

dx: enter a value for the subdivision of the x-axis

dy: enter a value for the subdivision of the y-axis

dz: enter a value for the subdivision of the optional depth axis. If set to 0 the subdividion is automatically determined from dy.

3.5 PlotGain/Filter

This group box allows to specify a gain or filter function for the plotting of the data.

AGCGain: if activated an AGC (AutomaticGainControl) with the given **Window** value (enter the wanted window length in the current timedimension) is applied for plotting the data. AGC facilitates the creation of equally distributed amplitudes in y-direction (normally time axis) within a predefinable window. This option serves for emphasizing of low amplitude ranges against ranges with high amplitudes. The information of the true amplitude is lost, of course. The program calculates at first an average amplitude over the total time range for each trace. After that the program scales each amplitude value in that way that the mean amplitude has the same value for each selected window around the current value with in a trace. The size of the window determines the kind of amplitude equality distribution. A window size of only one sample increment means that each time sample within one trace receives the same amplitude value (no reasonable choice), a window size of the whole trace length causes no modification of the amplitude. Small window sizes cause a strong equality distribution, large windows a weak.Often it is necessary to apply a scaling factor (parameter **Scale** with which the data are multiplied after the application of the AGCGain) smaller than one because after the application of the AGCgain some amplitude values will exceed the maximum amplitude of the original profile. The optimal choice of this scaling factor enables that all amplitude values will not exceed the limit of 16 bit.

EnergyDecay: if activated the energy decay curve is applied for plotting the data. By activating this option a gaincurve in y-(time-)direction is applied on the complete profile based on the mean amplitude decay curve. First a mean decay curve is determined from all existing traces. After the application of a median filter on this curve every data point of each trace is divided by the values of the decay curve. Often it is necessary to apply a scaling factor (parameter **Scale** with which the data are multiplied after the application of the EnergyDecay curve) smaller than one because after the application of the gain some amplitude values will exceed the maximum amplitude of the original profile. The optimal choice of this scaling factor enables that all amplitude values will not exceed the limit of 16 bit.

3.6 Plotsuboptions

This group box controls the main plot settings.

ShowMarker: if activated the distance markers are shown as white rectangles.

Commentmarker: if activated the comment markers are shown as yellow rectangles together with the comment.

TraceHeaderDistances: if activated the profile is plotted based on the individual distances stored in the single traceheaders and not based on the equal trace increment of the fileheader. The option is only available within the wiggle plotmode.

AutoInterpolation: if activated an autointerpolation in x- and y-direction is done for the pointmode. Activate this option if your data density is smaller than the plotting area (for example when you did a large zooming). Plotting takes more CPU time when this option is activated.

Rotate90Degree: if activated the profile is rotated by 90 degrees.

FlipYAxis: activate this option if you want that the y-axis starts at the bottom.

FlipXAxis: activate this option if you want that the x-axis starts at the right. With this option activated it is not possible to do some analysis steps like picking.

ShowAxis: if activated the x- and y-axis are plotted.

Grid: if activated a grid is plotted. The option grid color specifies the color of the grid.

XAxisName: activate this option for a manual labelling of the x-axis. This manual labelling is stored in the INI file after terminating the program and is loaded when starting the program. If the option is deactivated, the x-axis name string stored in the fileheader of the current line is used for the labelling.

YAxisName:activate this option for a manual labelling of the y-axis. This manual labelling is stored in the INI file after terminating the program and is loaded when starting the program. If the option is deactivated, the y-axis name string stored in the fileheader of the current line is used for the labelling.

AxisWithExponent: if activated the axis labelling is done with exponential representation if the labelling values exceed some predefined values (e.g. values between 10000 and 30000 are displayed as 100*10**2 and 300*10**2).
If activated and no manual axis labelling (options XaxisName and YaxisName deactivated) is entered the following holds true in addition:
timeaxis: automatic display in µs instead of ns and ms instead of µs and s instead of ms if the timerange is bigger than 10000.
distanceaxis: automatic display in KM instead of METER if the distancerange is bigger than 10000.

reduction velocity: if activated the traces are plotted with a timeshift calculated from the trace-distance and the entered velocity. The option is useful for e.g. displaying refraction data.

DepthAxis: if activated an additional depth axis is plotted. The depth axis is calculated from the timeaxis and the given velocity (see below).

v[m/ns]: enter a value for the velocity for the calculation of the depth axis.

Elevation: With the option elevation activated the depth axis on the right hand side is replaced by an elevation axis showing the elevations based on the entered ref. level. The current elevation is calculated from: reference level - current depth value.

correct header elevations: if activated the traces are shifted based on the receiver and the shot elevation values stored within the traceheader of each trace and the entered elevation level. The shift levels are calculated from the difference of the entered elevation level and the individual traceheader elevation values. Based on the current velocity the traveltime shift value is calculated from the sum of the shot and the receiver elevation differences.
To be considered: the range of the timeaxis will not be changed using this plot option. Therefore it can happen, that - depending on the chosen elevation level - information of some traces is not plotted because it is shifted to times outside of the time range.

depthaxisname: activate this option for a manual labelling of the depth-axis. This manual labelling is stored in the INI file after terminating the program and is loaded when starting the program. If the option is deactivated, the y-axis name string stored in the fileheader of the current line is used for the labelling.

showvelocity: if activated the depth axis labelling containes the velocity or the velocity file which are the base for time-depth conversion.

4. Print Menu

This menu allows you to setup the parameters for printing a profile. Batch printing is supported - see PrintOptions2.The scale in x- and y-direction is freely choosable (see also PrinterSize). The scale is entered either by direct input of the scale or by entering the length of the individual x- and y-axis. A page blocking option is included (see also PrinterSize and PrintOptions1). Printing on banner paper (printing on continuous paper) is supported.

Following you find a detailed description of the print options:

automatic center: if activated the profile is automatically centered on the sheet. Automatic centering is automatically disabled if printing is done in the Printing on banner paper mode (continuous paper).

X-scale output: the given scale is used for determining the x-size of the output.

Y-scale output: the given scale is used for determining the y-size of the output.
With depth axis activated (see also Plotsuboptions) the scale is applied on the depth axis and not on the time axis of the profile.

page blocking: if activated only one block of the given profile length/page is printed on one page. The program automatically subdivides the profile into several parts of constant length which will be printed on individual sheets.
Page blocking is automatically disabled if printing is done in the Printing on banner paper (continuous paper).

Landscape: if activated the output is landscape. With deactivated option the output is portrait. The current settings within the printer setup for landscape or portrait are overwritten.

use current zoom: if activated the currently set zoom parameters are used for the printing. If deactivated the complete profile(s) is plotted.

x-axis length[cm]: enter the x-axis length of the output in cm. If page blocking is activated, this length specifies the x-axis size of one page block.

x-scale[relation-1:?]: enter the scale if the option x-scale output is activated. For example: profile length 200 m. You enter a x-scale of 1000: the output has a length of 20 cm.

y-axis length[cm]: enter the y-axis (normally timeaxis) length of the output in cm.

y-scale[relation-1:?]: enter the scale if the option y-scale output is activated (see also x-scale[relation-1:?]).

upper border[cm]: enter the upper border in cm.

left border[cm]: enter the left border in cm.

profile length/page: enter the length of the profile part bo be printed on one page if the option page blocking is activated.

fast print: if activated a fast algorithm is used for printing the data for the point mode. In some cases dependent from the printer this printing method fails (only the frame is printed or the colors are not correct) and you must deactivate the option in order to print out the data.

print general comment: print a general comment - this comment is printed at the upper left corner of the image.

print frame: print a frame around the output. Print frame is automatically disabled if printing is done in the banner mode (continuous paper).

print filename: print the filename on the top of each profile.

print header boxes: if activated the program asks for the header box file to be loaded after having activated the option print. For a detailed description of the header comment boxes see chap. print preview.

Text Font: enter the font for the text of the output.

Number Font: enter the font of the numbers of the output.

Symbol Font: enter the font for the symbols to be printed (e.g. picks or markers).

Please use only true type fonts because only these fonts are able to be rotated.

Partitionscans: enter the max. number contained within one single print bitmap. The program automatically subdivides the printer bitmap based on the entered number. If problems occur with the printing (e.g. something is missing) it might be helpful to decrease this number. The number does not restrict the total number of scans to be printed.

PrinterSetup: Enter the printer-setup menu.

Preview: Enter the print preview menu which allows a preview of the size of the printoutput and to define text boxes (for a detailed description see chap. 4.2).

CANCEL: break off printing

PRINT: start printing

Printing on banner paper

Banner printing (printing on continuous paper) is supported. The following restrictions are valid for banner printing:

- no automatic center
- no page blocking
- no print frame
- the border in banner direction is always set to 0

- because banner printing is still a page based printing the following problems may occur when switching to the next page:
- axis numbering is not correct
- the wiggles may be disturbed at this position

4.1 Print preview

The print preview menu allows you to preview the size and shape of the print output and to define a **print heade**r consisting of up to 30 different header comment boxes containing up to 6 different comments. The print area is whitened and a simple preview of the profile(s) is shown within this area. To be considered: The profile preview is only a simple bitmap transform from the screen and does not exactly respresent the print output especially concerning the axis size. Some print options like print general comment or print file comment and print frame are also not taken into account.

If the printing is done onto **several pages** different print headers for each page may be defined (see option page (max.)). In this case the profile itself is not shown but you only may define the print headers.

The following general options are available:


resets the x- and y-scale values (zoomvalues) to 1 and replots the print preview


replot with current zoom parameters


enable manual zoom - With the option ZOOM an arbitrary area of the print preview can be selected and plotted in full screen size. The area to be enlarged, a rectangle, has to lie within a data set. Pressing the left mouse button you determine a corner of this rectangle and by moving the mouse with pressed button the desired area is opened. Afterwards you may use the scroll buttons to scroll through the zoomed print area.

close: close the print preview window without printing.
print: does printing using the current comment box settings and closes the print preview window.

reset act: clear the current comment box.

reset all: clears all comment boxes.

load: loads an existing set of comment boxes from file.

save: saves the existing set of comment boxes on file. The file will have the extension hea and is stored under the path rohdata under the current project directory. You must store the set of comment boxes if you want to use the boxes without entering the print preview window (see option print header boxes).

add: add header boxes (one additional file) to the current ones

page (max.): choose the wanted print page for defining the print headers

The **print header** is built up of up to **30 different header comment boxes** each containing up to **6 different comments**.

One **comment box** is characterized by the following values:

X-pos:enter the x-coordinate of the left corner in cm
Y-pos: enter the y-coordinate of the top corner in cm
width: width of the box in x-direction in cm
height: height of the box in y-direction in cm
degree°: specifies the rotation angle of the comment text - e.g. 0°: horizontal alignment, 90°: vertical

alignment
pen width: specifies the width of the box frame
frame: specifies the color of the box frame
fill: specifies the fill color of the box if transparent if deactivated
transparent: if activated the box is transparent, activate this option for example if you want to place some comments within your data
bitmap-file: if activated a bitmapfile can be loaded into the box, e.g. a logo.

The **actual box** is highlighted by a big frame. An existing box may be activated by pressing the left mouse button wihtin the wanted box. With pressed left mouse button the current box may be moved to any position.
The edit option **box nr.** shows you the number of the current box. This option also allows you to choose any of the existing boxes.

A **new box** is interactively defined by spanning up the wanted area. Click on the uppermost left corner of the area to be spanned up and drag the mouse to the wanted lowermost right corner with the left mouse button pressed.

Each comment box may contain up to **6 different comments**. Each comment is defined by the following parameters:
the comment text itself
font: enter the wanted font of the comment
X-Pos.: enter the x-position in cm within the comment box.You may use the up and down arrows for a fast replacement of the comment. The option change size defines the stepwise size of each redefinition.
Y-Pos.: enter the y-position in cm within the comment box. You may use the up and down arrows for a fast replacement of the comment. The opiton change size defines the stepwise size of each redefinition.

To be considered: A **horizontal** or **vertical line** is easily constructed by defining a new header box with a width (vertical line) or height (horizontal line) smaller than 0.05 cm.

5. Velocity adaptation

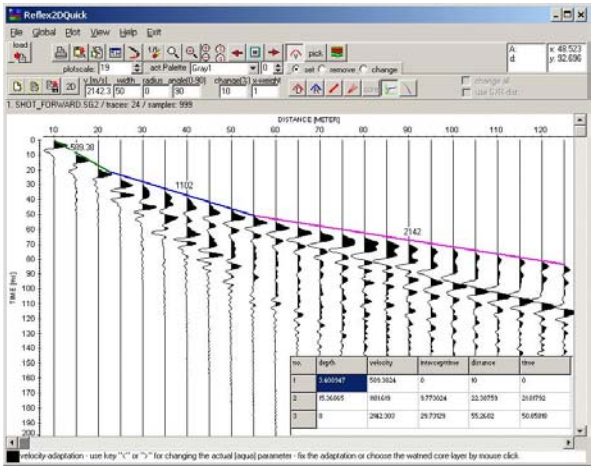
The option interactive velocity adaptation allows the interactive adaptation of **diffraction** or **reflexion hyperbola** by calculated hyperbola of defined velocity and width. There is also the possibility of fitting **linear features** either by interactively changing a line or by setting two points. 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.

You may choose between the adaptation of linear features (speed button  and speed button ), of diffraction hyperbola (speed button **D**, the profile must be a zero offset profile) and of reflexion hyperbola (speed button **R**) and of core data (speed button **core**).

In addition there is the possibility of an intercepttime inversion for refraction data (speed button ) as well as a direct wave adaptation of VSP-data (up or downhole data, speed button ).

With the speed button  the program waits for the input of two data points for generating a straight line of distinct velocity. Pressing the left mouse button determines the first data point and by moving the mouse with pressed button the line with the current velocity is shown. Leaving the mouse button holds the current line.

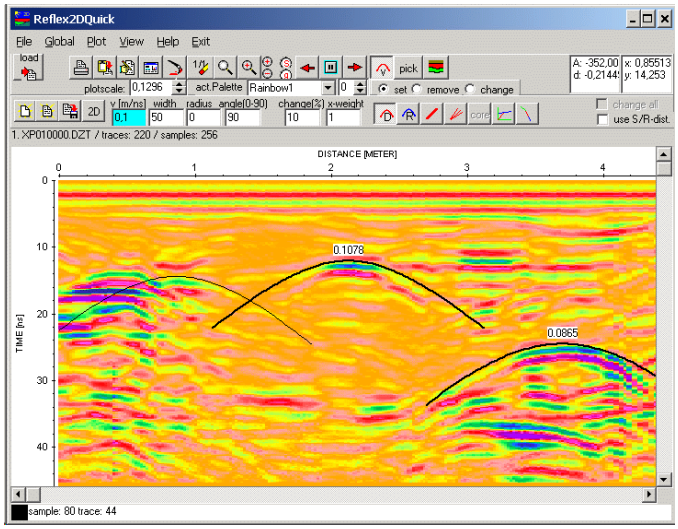
The speed button  allows the interactive use of the intercept time method for refraction data. The option enables to get a first 1D-model very quickly. After having activated the option you must enter the bending points by the left mouse button. The first point is automatically set to time zero and distance of the shot position. Move the mouse with pressed left mouse button to the next bending point and leave the mouse button and so on. The start point of a new branch always corresponds to the endpoint of the previous one. The velocites derived from these settings must increase with depth. After having finished the settings the depth and velocities of the calculated 1D-model are shown in a window. Using the option save  (file with extension HYP) an ASCII-1D-model-file with the extension VEL is automatically saved in addition, which allows an easy access to the interpreted 1D-model.



The speed button  allows the interactive adaptation of the direct wave of a VSP (vertical seismics profiling) measurement (uphole or downhole measurements). After having activated the option you must enter the bending points by the left mouse button. The first point is automatically set to time zero and distance of the shot position (normally 0). Move the mouse with pressed left mouse button to the next bending point and leave the mouse button and so on. The start point of a new branch always corresponds to the endpoint of the previous one. After having finished the settings the depth, velocities and thicknesses of the calculated 1D-model are shown in a window. Using the option save  (file with extension HYP) an ASCII-1D-model-file with the extension VEL is automatically saved in addition, which allows an easy access to the interpreted 1D-model.

The speed button **core** is only enabled when a core datafile is loaded (see suboption Core data/1D

models under View MenuItem). 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 < or > respectively (see below). With the option **change all** activated all layervelocites of the actual cores are changed simultaneously.



With any of the other speed buttons activated a hyperbola or a line according to the adaptation type based on the current parameters is calculated and plotted, as soon as the cursor is within the working window of the profile.
Important:
The type **reflexion** adaptation needs CMP- or Moveout-data.
The type **diffraction** needs zero-offset data or a negligible source-receiver distance with respect to the depth of the object. With the option **use S/R-dist.** activated the source-receiver distance stored within the fileheader is taken into account for the calculation of the diffraction hyperbola. This mainly affects the apex of the hyperbola.
By pressing the left mouse button the current diffraction hyperbola is maintained and plotted on the profile. Additionally the current velocity is displayed in a window.

All the adaptation parameters **v**, **width**, **radius** and **angle** can be interactively changed using the keys < or > respectively. The parameter which can be changed is shown in aqua color. Activate the parameter to be changed interactively by simply enter it. The size of the interactive change is controlled by the parameter **change(%)** given in percentage.

The input parameters in detail:

v(all): input of the starting velocity for the interactive adaptation (input in m/ns or m/s).

width(diffraction, reflection, linear adaptation): input of the one-sided width of the hyperbola to be calculated in traces.

radius(diffraction hyperbola-adaptation): input of the presumed radius of the target object (specification in the dimension of the current profile, input 0 for point-shaped object).

angle(diffraction hyperbola-adaptation): input of the presumed angle between the profile direction and the direction of the line target. Input in degrees between 0 and 90 degrees. For a point-diffractor you have to enter an angle of 90 degrees.

x-weight (all): input of a weight parameter for the 2-dimensional velocity interpolation. The interpolation is done as following: all current velocities are summed up for every point in the x-t range according to the square of their distance to the x-t-point. A value for x-weight greater than 1 favours the x-direction (line direction). By choosing a value big enough you may get a vertical layering based on only few velocity values.

The speed button on the left-hand side allows to **reset** all adaptations, to **load** existing velocity

adaptations and to **save** the current adaptations on file with the extension HYP under the rohdata directory.
Using the option **save** it is also possible to save the adaptations using an **ASCII-format** with the extension ASC. Each row contains the distance, velocity and time of one adaptation. In case of the intercepttime adaptation each row contains the calculated depth, the velocity, the intercepttime, the distance and the time. In case of the VSP adaptation each row contains the calculated depth, the velocity, the thickness and the time.
The loaded adaptations are plotted at their original profile distance. This means that you have to be careful if you are using adaptations from other, e.g. crossing profiles. When saving the current velocity adaptation, the following parameters are stored: velocity, distance in profile direction, x-weight.

Peculiarity for speed button **core**: With the option save corefile under global settings activated the currently loaded corefile is updated using the current layer velocities. With the option save VLA-file under global settings activated a VLA-ASCII velocity datafile is automatically created.

Using the option **2D** it is possible to show the 2-dimensional velocity distribution based on the current adapted velocities. The interpolation is controlled by the parameter x-weight.

The options **set**, **remove** and **change** control the settings of a line or a hyperbola. With the option set activated a new adaptation may be set using the left mouse button. Any adaptation may be removed by activating the option remove and choosing the desired adaptation.

The max. number of different adaptations is 100.

6. Pick Menu

With the pick option activated arrivals can be picked. You have the choice between manual picking, continuous pick and a semi-automatic picking using a phase follower.
After having activated the pick option you still have the possibility to activate the velocity adaptation option (chap. 5) in addition. Thereby for example it is possible to continuously display a calculated diffraction hyperbola during picking for a better determination of the hyperbola cusp.

Corrections can be performed either on zero crossings or on the extremum (option CORRECTION). The picks can be saved with an arbitrary name and reloaded any time (option save and load). The maximum number of picks is limited to the maximum number of scans per profile. If you need more picks for a profile, they must be saved on different files.

Both the travel time and the current amplitude of the phase are detected. The manual picking of phases is superior to the automatic picking in respect to time consumption for small data sets and in respect to accuracy for data sets of bad quality. For large data sets it is recommended to have a desired phase semi-automatically picked, first - this is discussed under phase follower, and then to check the quality of the automatically set picks and correct them manually if necessary.
In addition to the distance the xyz-shot- and receiver coordinates stored in the traceheader (see also trace header Edit menu, chap. 2.2) of each individual trace are stored within the pick record. These values are used when using the tomography or when exporting the picks to an ASCII-file e.g. in order to display a 2- or 3-dimensional contour plot of the pick values. If you want to create such an ASCII-file you must ensure that the xyz-shot and receiver coordinates are correct. These traceheader-coordinates may not always correspond to the fileheader-coordinates and therefore you have different possibilities to update them (see trace header Edit menu, chap. 2.2).

Within the global settings menu you can choose between two different symbols for the picks (* and -). Here also some other pick paramters can be entered. The size of the pick-symbol can be changed within the symbol font menu (see also Global MenuItem).

manual pick: With the option **set** activated a pick can be set with the left mouse button. The same applies to the manual removal or change of an existing pick after selecting the options **remove** or **change**, respectively. Here those picks are removed or changed which are next to the current cursor position.The right mouse button activates the interpolate option whereby a fast piecewise linear picking is possible.

continuous pick: With this option activated you can set (option set activated), remove (option remove activated) or change (option change activated) picks by moving the mouse cursor over the data set with continuously pressed left mouse button. The picks are set or removed or changed at those traces that are next to the current cursor position. The continuous picking must be done from left to right, this means with increasing distance coordinates. The option continuous pick therefore makes manual picking a lot easier for large data sets. If the cursor movement is too fast to pick every trace an interpolation between the founded pick position is done. The parameter delay under global settings controls the speed of the continuous picking. The data must be sorted in ascending order with respect to the distances (see also option sort).
To be considered: The continuous picking is done based on the tracenumbers and these tracenumbers are stored with the picks. Therefore problems may occur when you are loading existing picks which have been picked from another dataset with different tracenumbers (this means with different traceincrement and/or startcoordinate).
The right mouse button allows you to toggle between the options set and change.
With the plotoption TraceHeaderDistances activated no interpolation is done.

phase follower: The option allows the automatic assignment of picks to a selected phase. For this you first have to select the desired phase at an arbitrary position of the profile with the left mouse button. With selected secondary option **set** the program then follows automatically this phase in the direction of larger distances. The choice is limited to the displayed screen range. If the phase follower reaches the right end of the screen the profile is scrolled automatically towards larger distances and the pick assignment is not stopped (the option set is also automatically set to change). A precondition for a correct application of this option is that the phase extends continuously over the whole profile without being perturbed by interference effects. The automatically detected picks certainly can be reedited like all other picks.
If the option **change** is activated already existing picks can be changed with the automatic assignment. The program expects like with selected option set the selection of a value within the profile by pressing the left mouse button. If for the selected trace already a pick exists it is replaced by the new automatically assigned pick. With this option it is possible to react fast on an unwanted deviation of the automatic assignment from the desired phase.
The Escape-key as well as the right mouse button allows to abort the automatic assignment.
The speed of the phase follower is controled by the option **delay** (available within the PickGroupBox if the option **add.options** is activated - see below). Increase this number in order to slow down the phase follower.

Auto pick: With this option activated the Auto-Pick window opens which allows you to set the autopick parameters and start the automatic picking (see also chap. 6.1).It is possible to bring this window to the front by pressing the right mouse button within the profile.

difference pick: With this option activated the traveltime difference between two successive picks is automatically determined. In addition the thickness is calculated based on this traveltime difference and the entered velocity (see PlotOptions chap. 3). First you must enter the reference pick and then the investigation pick. The two values traveltime distance and calculated thickness are plotted right to the investigation pick using the number font if the option show pickdifferences within the global settings menu (also active when printing the profile) . The number of total places and decimal places can be defined within the global settings menu. The remove button as well as the sort and control buttons are disabled. After having activated this option all picks are reset.
With the option keep distance within the global settings menu activated the x-position between the two picks are plotted in addition. With this option not activated the x-position of the second pick is automatically moved to the x-position of the first one.

pick code: input of a code (integer value) for the characterization of the following picks to be chosen. The picks with the current code are plotted using the current set pick color (see option color below). Picks with another code than the current one are plotted using either the layershowcolor corresponding to their code or the color defined within the option panel "not active picks" within the add. options (see below). When changing an existing pick the newly set pick always obtains the current code. With the option **use code** activated only the picks with the current code can be changed or removed.

use code: if activated the pick code is used for discriminating the individual picks:
- only the picks with the current code may be changed or removed
- different colors are used for the picks with different codes (with the option use layershow-col. activated the layershow colors are used)
- options correct, auto cor. and control only act on the picks with the current code
- when using the phase follower the option use code should be activated if several reflectors at the same distance are picked.

if deactivated the following holds true:
- all picks may be changed or removed

- the current pick color is used for all picks
- one correction type is used for all picks

color defines the color of the picks with the current code. The color of the picks with other codes and deactivated option use layershow-col. is defined within the add. options menu (panel not active picks).

use layershow-col.: if activated the layershow colors are used for the pick display corresponding to their code.

load: This option allows to load new picks from disc. Please note that the current picks are lost with the loading of new picks. After selecting the option you can choose one or several datafiles from the file list.
It is possible to load REFLEX formatted PCK files as well as different ASCII-formats:
ASCII-2-colums: each line of the ASCII-file corresponds to one pick and consists of two columns containing the distance and the time in the corresponding dimensions.
ASCII-3D-tomography: each line containing the following values:
traveltime code x-shot y-shot z-shot x-receiver y-receiver z-receiver
ASCII-colums: each line contains the following informations:
distance offset(profileconstant) x-shot y-shot x-rec y-rec traveltime depth amplitude code
When loading the Reflex formatted PCK files the program automatically controls if the tracenumbers stored with the picks correspond to the profile tracenumbers. If not a warning message appears and a query if the pick tracenumbers shall be updated. The tracenumber are used for continuous picking and for the phase follower. Therefore the tracenumbers should be updated if one of these options shall be used.

reset: This option allows to reset all current picks. The current pick settings are lost.

sec.picks: loading of picks already saved, which are plotted as comparative picks in addition to the currently set picks. You may choose different pick files from the file list. The total number of secondary picks must not exceed the max. number of scans. The color of these secondary picks may be freely chosen using the "color sec.pick" button within the add.options or it is automatically determined from the layernumber and the associated layershow colors (option "use layershow-colors" within add.options activated). After deactivating the option sec.picks the secondary picks are reset. The secondary picks are only for comparison and are not saved together with the current picks.The secondary picks are also shown on the second profile.

save: saves the current picks. After selecting the option the Pick save menu will be opened.

autom.load: if activated the picks stored under the same filename like the currently loaded file (see also option automatic name under Pick Save MenuItem, chap. 6.2) are loaded automatically.

sort: sorts the picks with ascending order with respect to the distance axis. The continuous picking needs sorted data. Activate this option if the continuous picking is not working. The data are automatically sorted after having saved.

correct: activate this option to correct the current picks. With the option use code activated only the picks with the current code are corrected for. The following correct options are available:

correct to max.: allows to correct the current picks to the extremum of the signal belonging to the pick.

correct to zero: allows to correct the current picks to the zero crossing of the signal belonging to the

pick.The zero crossing will not only be done to the nearest data point but to the interpolated values between the two datapoints of the zero crossing.

corr.max/min.time:allows to correct the current picks to the extremum of the signal and the minimum time within a tracerange of a predefined number of traces (see input pickcorrect traces under global settings). Use this option for example to correct the picked cusp of a diffraction hyperbola.

corr.max/dist.range: allows to correct the current picks to the extremum of the signal within a given tracerange of a predefined number of traces (see input pickcorrect traces under global settings). The timerange for searching the maximum is automatically restricted by the polarity of the current pick. Use this option for example to correct the picked cusp of a migrated diffraction.

auto cor.: if activated the picks are automatically corrected during the setting. The above correct options are valid.

control: this option allows you to remove double picks, this means several picks for one distance (one trace). Double picks may occur if you are using the continuous picker with activated set option and you don't start the picking at that position where no picks have been set yet.

interpolate: allows the linear interpolation between the two last picks or the extrapolation to the profile start and the profile end if only one pick exists.

The four **arrows** at the right side allow to move up and down and to the left and to the right the last set pick by one sample (trace) increment.
Activate the option **all** to move up and down (to the left or to the right) all picks by one sample (trace) increment. With the option **use code** activated in addition only the picks with the current code are moved.
To consider when moving the picks to the left or the right: the xy-coordinates of the picks are not updated when using this option. This must be done during saving the picks with activated option xy-coordinates.

The option **add. options** opens an additional PickGroupBox containing the following options and the options autostop autostart and last pick stop already described under phase follower (see above):

The parameter **n.trace** determines that only for every n. trace a pick is found for the continuous picking and for the phase follower (the phase follower however acts on every trace but only every n. trace a pick is chosen).

max.diff: This parameter specifies a stopping criterion for the phase follower. The time difference between successive picks of the same polarity must not be larger than maxDiff*Time increment. The default value of maxDiff is 5. You should reduce this value if the phase is lost too often. You have to increase this value if the phase does not follow the extremum.

marker picks: allows to automatically set picks at the existing marker positions. The time values are set to 10*time increment. The option is only available if the markers are stored in the trace header.

load perpendicular picks: allows to load and to correct existing picks for the xy-coordinates stored within the traceheader of the current datafile (see also trace header Edit menu, chap. 2.2). Those picks which xy-coordinates do not correspond to any existing xy-coordinate of the current datfile are automatically cancelled. The option might be useful if existing picks are loaded into a crossing line. The parameter **corr.xy-coord bin(%)** under global settings controls the size of the crossing bin. Increase this parameter if the option control xy-coord does not generate any picks.

remove non double picks: allows to remove all picks for which only 1 pick for one distance position exists. The option is useful for a continuous picking of a layer thickness. For that purpose first pick the upper border of the layer and then the lower border. Then use the option remove non double picks in order to ensure to have 2 picked values for each location. Then you may use the ASCII-pick difference output within the Pick Save MenuItem (chap. 6.2) in order to output the thickness of the picked layer.

change pick code: allows to change the codes of the picks with the current code. After having activated the option the new code is queried. The option is only enabled if use code is activated.

move based on sec. picks: Activating the option moves the current picks to larger times based on the loaded secondary picks. The option might be used for example for an adaptation of existing pick files to a topographic correction. For this purpose the interpretation flow is the following:
a. pick the reflections based on the non topographic corrected profile
b. perform the topographic correction
c. pick the first arrival within the topographic corrected profile
d. load these first arrival picks as secondary picks
e. load the wanted pick file and activate the option “move based on sec. picks” - the picks will be adapted to the topographic correction.
To be considered: Moving is done based on the times of the sec. picks - therefore the starttime of the profile should be 0. If this does not hold true or if you are not able to pick the very first arrival you still have the possibility to move the surface reflection picks up or down using the red arrows with activated option "all".

show pick-code: if activated the pickcode is shown next to the pick

code auto-increase: if activated the pickcode is automatically increased when a pick is manually set

code transparent: if activated the display of the pick codes or the pick differences will be transparent

not active picks: within this panel you may specify the **color** for the picks with another code than the current one (see also option pick code) in the case the option use layershow-col. (see above) is deactivated. The size and the shape of the picks are determined from the current settings of the symbol font.

sec.picks: within this panel you may specify the **color** for the secondary picks. With the option **use layershow-col.** activated the layershow colors are used for the pickdisplay corresponding to their code (option use code activated) or corresponding to the given layernumber (option use code deactivated).

6.1 Auto-Pick MenuItem

Within this menu you have the possibility to automatically pick the wanted onsets.

In any case it is recommended to check the automatic picks. For that purpose you may use all the other existing picking possibilities described above.

It is possible to bring this window to the front by pressing the right mouse button within the profile (phase follower and continuous pick must be deactivated).

At present it is possible to pick the following objects (option **object type**):
single objects like small diffractions (e.g. from rebars) - data should be migrated
first arrivals (e.g. from refraction seismics)
continuous reflector

The following parameters are valid for all object types:

start time: enter the start time of the time range for detecting the objects.

end time: enter the end time of the time range for detecting the objects.

total distance range: if activated the complete distance range is valid for detecting the objects.

start dist.: with deactivated option total distance range enter the start distance of the distance range for detecting the objects.

end dist.: with deactivated option total distance range enter the end distance of the distance range for detecting the objects.

slope °: enter the slope angle in degrees for determing the time/distange range for the automatic search of the picks. With a value <> 0 the start and end time for the detection are changing with distance (distance*tan(slope)). A value <> 0 allows you for example to automatically detect a continuously dipping reflector.

Polarity: enter the polartiy of the onsets to be detected.

interpolate: if activated a linear interpolation between the detected picks will be automatically performed. If only one pick was detected an extrapolation to the profile start and the profile will be done.

CorrectBox: within this box you may determine how the picks are corrected after having detected:
no: no correction is done.
max.: allows to correct the current picks to the extremum of the signal belonging to the pick.
zero: allows to correct the current picks to the zero crossing of the signal belonging to the pick.
max/min.time: allows to correct the current picks to the extremum of the signal and the minimum time within a tracerange of a predefined number of traces (see input pickcorrect traces under global settings). Use this option for example to correct the picked cusp of a diffraction hyperbola.
max/dist.range: allows to correct the current picks to the extremum of the signal within a given tracerange of a predefined number of traces (see input pickcorrect traces under global settings). The timerange for searching the maximum is automatically restricted by the polarity of the current pick. Use this option for example to correct the picked cusp of a migrated diffraction.

Depending on the object type you must enter different auto-pick parameters.

Object type **single objects**:

A single object is dermined if the following criteria are fullfilled:

1. The polarity must fit.
2. The onset must be located within the given time/distance range.
3. The amplitude must exceed a given threshold.
4. The amplitude contrast within a given distance range must exceed a given threshold.

The two threshold parameters are automatically calculated from the automatically determined mean values and the given scaling factors.

amp.scale: enter a scaling factor for determining the amplitude threshold. The program automatically calculates the mean amplitude value within the given time/distance range. The threshold is determined from this mean value multiplied by the amp.scale factor.

contrast scale: enter a scaling factor for determining the amplitude contrast threshold. The program automatically calculates the mean amplitude contrast value within the given time/distance range based on the parameter xsize. The threshold is determined from this mean value multiplied by the contrast.scale factor.

xsize: enter the expected size in x-direction of the objects. This value is used for the determination of the amplitude contrast as well as for excluding picking one object twice on different traces or samples.

ysize: enter the max. expected size in y-(time-)direction of the objects. The polarity must remain the same within the given y-size. Otherwise the pick will be rejected.

x-interval: enter the x-range behind a detected pick where no pick detection will be done.

Object type **first arrivals**:

A first arrival is dermined if the following criteria are fullfilled:

1. The polarity must fit.
2. The onset must be located within the given time/distance range.
3. The amplitude must exceed a given threshold.
4. The instatnteneous frequency must be in the range of the mean instantaneous frequency. The range is given by the mean instantaneous frequency and the frequency scale parameter.

The two threshold parameters are automatically calculated from the automatically determined mean values and the given scaling factors.

amp.scale: enter a scaling factor for determining the amplitude threshold. The program automatically extracts the max. amplitude for each trace within the given time range and at the given polartiy. The threshold is determined from this max. value multiplied by the amp.scale factor.

frequency scale: enter a scaling factor for determining the range of the allowed instantaneous frequencies. A factor of 1 means that a 100 % deviation from the mean instantaneous frequency is

allowed. Accordingly a factor of 0.1 means 10 % deviation and a factor of 10 1000 % deviation. The program automatically calculates the mean instantaneous frequency for all traces within the given time/distance range.The threshold range is determined from this mean value multiplied by the frequency scale factor.

Object type **continuous reflector**:

A continuous reflector is automatically picked at those positions where:

1. the polarity fits.
2. the max. amplitude value within the given timewindow is given.

default: sets some default parameters for the types single objects and first arrival.

The parameters within the control panel box are the following:

save par.: save the current parameters on file with the extension api.

load par.: load the auto-pick parameters from file. The fileextension must be api.

start: start the autopicking for the current file. The picks will receive the currently set color and code and are displayed after having been detected.

batch start: start the autopicking for a choosabele number of files.
After having activated this option you must select the wanted profiles from the filemenu. Afterwards the automatic picking is started for all chosen profiles. The detected picks of the different profiles are stored on individual files using the ReflexWin pickformat (extension PCK under the path LINEDATA) and the filename automatically determined from the acutal profile filename (see also option automatic name under Pick Save MenuItem, chap. 6.2).

close: close the auto-pick window.

6.2 Pick save MenuItem

Save the current picks using different formats.

Control Panel section:

filename: specify the filename. Based on the chosen formats the file will have different extensions and will be stored under different directories:
ReflexWin: extension PCK and storing under the path LINEDATA under the program directory.
ReflexDOS: extension TRV (travel times) and AMP (amplitudes) under the path LINEDATA under the program directory.
ASCII-columns: extension PCK and storing under the path ASCII under the program directory
ASCII-tomography: extension TOM and storing under the path ASCII under the program directory

automatic name: if activated the file name is automatically determined from the file name of the current profile.

backup file: if activated an additional pickfile with the name ???_backup.pck will be created.

save: the option starts the storing of the data. For the output formats ASCII-columns and ASCII-tomography the created file output is displayed within a list box. Closing this box terminates the window.

cancel: close the window without saving the picks.

FormatGroup Section:

The following output formats are supported:

ReflexWin: new binary REFLEX pick format containing both the travel times and the amplitudes - data cannot be loaded with the DOS-version of REFLEX anymore.

ReflexDOS: old binary REFLEX pick format. Travel times and amplitudes are stored on different files with the extensions TRV and AMP.

RefraDOS: old binary REFRA pick format. Traveltimes and amplitudes are stored on a file with the extensions TRV under the directory TRAVTIME.

ASCII-columns: convert picks to an ASCII file whereby every pick uses one line.
With the option **export several existing picks into 1 ASCII-file** activated you may export several existing pickfiles into 1 ASCII-file. Each line of the ASCII-file contains the following values: traveltime(12:6), depth(10:4), amplitude(yy:xx), x-shot(yy:xx), y-shot(yy:xx), z-shot(yy:xx), x-receiver(yy:xx), y-receiver(yy:xx), z-receiver(yy:xx) where yy defines the number of places and xx defines the number of decimal places given within the global settings menu (chap. 1.2.1). The x-,y-,z-shot and receiver coordinates have been taken from the traceheaders of the individual profiles when picking the onsets (see also trace header Edit menu - chap. 2.2). Depths and amplitudes are set to zero if the corresponding options depth and amplitudes are deactivated.
With the option export several existing picks into 1 ASCII-file deactivated it is possible to save the existing picks into an ASCII-file. every pick uses one line. Each line contains the following values: coordinate in profiledirection(10:3), coordinate in profileconstant(10:3), x-traceheadercoordinate (optional,yy:xx), y-traceheadercoordinate (optional,yy:xx), travel time(12:6), depth(10:4),

amplitude(yy:xx) where xx defines the number of decimal places given within the global settings menu. The number of total places can also be defined within the global settings menu. By default the travel times and also the plane-coordinates of the picks are written on a file. Additionally the travel time can be converted into depths on the basis of a specified constant velocity (option **depth** activated) and the current amplitude of each pick can be saved (option **amplitudes** activated). If no depths and/or amplitudes shall be extracted, these values are set to zero. Optionally the xyz-coordinates of the receivers (specification **xyz-rec.coordinates**) stored in each trace header may be written out (see also trace header Edit menu - chap. 2.2). This allows you for example to write out non equidistant xyz-coordinates or the xy-coordinates, if the profile is not orientated into one direction (x or y). The xyz-coordinates are only saved on file if chosen. With the option **xyz-shot/rec.coord.** activated both the shot and the receiver xyz-coordinates are written out. Optionally the acquisition times of each trace stored within the traceheader is written out (option **acquisition times** activated). Optionally the pickcode of each pick can be written out (option **pickcodes** activated).
The parameter **n.pick** controls the pick number to be output: only every n. pick will be output. The file has the extension PCK and will be stored under the path ASCII under the current projectpath.
With the option **original datafilename** activated the original datafilename (without path) is output in addition within the first column.

Ascii-2D tomography: every pick uses one line. It is possible to save the picked **traveltimes** or the **amplitudes**. Each line contains the following values: traveltime(yy:xx) or amplitude, code(10:0), x-tracehadercoordinate of the transmitter, y-traceheadercoordinate of the transmitter(yy:xx), x-traceheadercoordinate of the receiver(yy:xx), y-traceheadercoordinate of the receiver(yy:xx) where xx defines the number of decimal places given within the global settings menu (chap. 1.2.1). The number of total places can also be defined within the global settings menu. The xy-coordinates are taken from the corresponding traceheaders of each trace (see also trace header Edit menu - chap. 2.2) during picking. The file has the extension TOM and will be stored under the path ASCII under the current projectpath. The format corresponds to that format needed by the 2D-tomographic traveltime-inversion and amplitude-inversion.The distancedimension is always METER, the timedimension is either ms or ns. If the timedimension of the original data is µs, the traveltimes are automatically transformed into ms-data for storing.

Ascii-3D tomography: every pick uses one line. Each line contains the following values: traveltime(yy:xx), code(10:0), x-tracehadercoordinate of the transmitter, y-traceheadercoordinate of the transmitter(yy:xx), z-traceheadercoordinate of the transmitter(yy:xx), x-traceheadercoordinate of the receiver(yy:xx), y-traceheadercoordinate of the receiver(yy:xx), z-traceheadercoordinate of the receiver(yy:xx) where xx defines the number of decimal places given within the global settings menu. The number of total places can also be defined within the global settings menu. The xyz-coordinates are taken from the corresponding traceheaders of each trace (see also trace header Edit menu - chap. 2.2) during picking. The file has the extension TOM and will be stored under the path ASCII under the current projectpath. The format corresponds to that format needed by the 3D-tomographic traveltime-inversion. The distancedimension is always METER, the timedimension is either ms or ns. If the timedimension of the original data is µs, the traveltimes are automatically transformed into ms-data for storing.

GeoTomoCG: this format should be used when the picked data are interpreted using the tomography program GeoTomoCG.
2 comment lines are at the beginning. Then every pick uses one line. Each line contains the following values: pick number (5:0), x-tracehadercoordinate of the transmitter (13:3), y-traceheadercoordinate of the transmitter(10:3), z-traceheadercoordinate of the transmitter(10:3), x-traceheadercoordinate of the receiver(10:3), y-traceheadercoordinate of the receiver(10:3), z-traceheadercoordinate of the receiver(10:3), traveltime(10:3). The xyz-coordinates are taken from the corresponding traceheaders of each trace during picking. The file has the extension 3DD and will be stored under the path ASCII

under the current projectpath.
Activating the option **store y on z-coord** allows you to store the y-traceheader-coordinates on the z-coordinates of the GeoTomoCG-file.The y-coordinates of the GeoTomoCG-file will be set to 0.

ASCII-pick difference: allows to output the traveltime difference of two successive picks together with a calculated thickness. The picks may not be located at the same distance position. The pick pair is defined from the ascending order in distance direction. Therefore if you are using the manual or continuous pick for defining a layer thickness you should use the option remove non double picks (see Pick MenuItem, chap. 6) in order to ensure that the correct two picks are always used for calculating the pick difference.
Every pick pair uses one line. Each line contains the following values: distance for pick 2, distance difference, thickness, traveltime difference, traveltime 1, traveltime 2, Pick Code. The number of total places and decimal places can be defined within the global settings menu. The thickness is calculated from the traveltime difference and the entered velocity. With the option **xy-coordinates** activated the xy-shot and receiver coordinates of the second pick are written out in addition. The xy-coordinates are taken from the coordinates stored in each trace header (see also trace header Edit menu - chap. 2.2) of the current profile.
By default the picks are stored under ASCII under the current projectdirectory. Using the option **path** it is possible to enter another path. Then the filemenu opens. If no filename has been specefied before you must enter such a name. To be considered: it is still necessary to activate the option save in order to save the picks.

ReflexWin/ReflexDos section:

global code: A code is queried which serves for differentiating picks of different phases from one data set. Like this signal arrivals of compression waves can be marked with a P and the ones of shear waves with a S.

mean or layer velocity: Additionally a velocity can be preset which is used for the conversion of the travel times in depths under the option layershow. You have to specify a mean velocity for the total overburden or a layer velocity. Within the layershow you may specify whether mean or layer velocities are used for the time-depth convesion (see chap. 6.3.1).

layer number: The specification of the layer number serves to identify the individual picks within the option layershow. If you are using the option layer-show with the option pick, you therefore should process together and save on a file only those arrivals that belong to one layer.

check source/receivers positions: If activated the source/receiver positions of each pick are updated from the currently loaded profile. The source/receiver positions are used within the ASCII-export with activated option export several existing picks into 1 ASCII-file.

save only pick with actual code: if activated only those picks with the actually set code are saved

6.3 LayerShow

This option offers the possibility to combine individual pick files stored on file (see option PICK), 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 samples per distance point and thus of layer boundaries is 100.
After selecting the option you have the choice between

create - i.e. to create a new layer show from pick files and save them on file
load - load an existing layer show
create report - i.e. to create a reportfile based on the current layer show.

After selecting the option the screen is split automatically (Ver.split activated). In the upper window the profile is plotted, in the lower layer window the depth-converted combined arrivals (if already created with the option CREATE or loaded from hard disc) are plotted as continuous layer boundaries. The maximal displayed depth of the lower window is calculated from the maximal time of the upper window and a given **velocity**.

The option **x-flip** allows you to flip the current layershow in x-direction. The flipped layer-show is saved under a new filename.

If the option **show picks** is activated, additionally in the upper window the combined travel times (not depth-converted) are plotted as continuous layer boundaries, so that a direct comparison between raw data and picked arrivals is possible.
With the option **dist.axis** activated the distance axis is plotted for the lower layer window.
With the option **grid** activated a grid is plotted within the lower layer window.
With the option **fill layers** activated the layer above the boundary is filled using the current layershow color. The option is also available for printing. The color for the lowermost layer underneath the lowermost boundary is always white. To be considered: if the boundary is not continuous the color of the lower boundary will be used.

The current codings of the individual boundaries (see global code wihtin the Pick save MenuItem, chap. 6.2) are displayed based on the settings within the **legend** box using the symbol font. The colors of the codings are automatically determined from the layer numbers.The option **legend name** allows you to define the default prefixname of the legend - if this name is set to nil no legend is plotted. The size of one coding is calculated from the option **size** (enter the size of one coding in number of characters). The option **rows** defines the number of rows the legend is subdivided into. You have the following possibilities for defining the **legend position**:
Above means plotting the codings in the free space between the two windows.
Upper left means plotting in the upper left corner of the layer window.
Lower left means plotting in the lower left corner of the layer window.

Activating the option **colors** allows to change the plot colors and pen-thicknesses of the individual layer boundaries. A window appears where the predefined colors and the pen-thicknesses of the individual layer boundaries can be changed. The option save allows to save the layershow colors on a file with the extension LCO. The option load allows to load an existing layershow-colorfile.

6.3.1 Create LayerShow

This menu allows to create a file of combined equidistant picks. The "Input-picks", which do not have to be equidistant, have to be saved on file before under the option PICK. With choosing the option a window is opened with the following input or selection possibilities:

filename: allows the input of an arbitrary name. The combined picks automatically receive the extension LAY and are stored in the directory LINEDATA.

automatic name: if activated the filename of the layershow corresponds to the name of the currently loaded profile.

old format: if activated the old layershow format (until version 2.1.2 of REFLEXW) is used (max. 10 different layers). The new layershow format cannot be read from older versions of REFLEXW.

pick files: allows to load the different pick files. The maximum number of selectable files is 200.

start position: defines the start-position for the layershow file (has not to be identical to the start coordinate of the current loaded profile). At the beginning this coordinate is set to the start coordinate of the current loaded profile.

end position: defines the end-position for the layershow file (has not to be identical to the end coordinate of the current loaded profile). At the beginning this coordinate is set to the end coordinate of the current loaded profile.

increment: defines the increment between successive positions for the layershow file (has not to be identical to the trace increment of the current loaded profile - at the beginning this coordinate is set to the trace increment of the current loaded profile). For the combination of the picks the original distance value is modified by a multiple of this increment for a corresponding distance. For example you choose an increment of 0,1 METER and have picks at 10,24 and 10,31 METERS. The program sets these picks at 10,2 and 10,3 METERS for the combination. If for particular distances no picks exist, each time a space is inserted in order to allow that a layer boundary disappears at distinct regions. For the plotting of the combined picks in this case the picks to the left and right are not connected with a straight line. You should therefore choose the increment not smaller than the average distance between two picks.

velocity choice: specifies how the time-depth conversion of the picks is done. You have the choice between mean pick velocity and layer pick velocity.

Choosing **"mean pick velocity"** 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 pick velocity"** 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.Terefore the time-depth conversion of each layer boundary depends on the conversion of the upper boundary (see also option interpolate layer).

interpolate layer: option is used if the velocities are read from an ASCII-file (see option

VELOCITIES/velocity file) or if the option layer pick velocity has been chosen. The option controls how the time-depth conversion for a special layer is done if any upper layer is not continuously picked.
If the option is TRUE every upper layer is assumed to be continuous even if it is not picked. In this case 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 current layer is done. Activating this option is useful if you may assume that the upper layer is continuous but hardly to interpret in the reflexion data. The interpolation or extrapolation is done over the complete distance range except the option **< comments >** is activated. If this option is activated an upper layer will only be interpolated or extrapolated within those comment marker distance ranges where it really exists. It will be neglected within those comment marker distance ranges where it has not been picked.
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 current layer at the point where any of the upper layers is broken. Deactivating this option is useful for the case of a vanishing upper layer.

< comments >: if activated the interpolation or extrapolation of an upper layer is only done within those comment marker distance ranges where the upper layer really exists. The option is useful for example if the comment markers define construction changes with different layering. A comment marker distance range is defined between two successive comment markers and from the profilestart to the first marker and from the last marker to the end of the profile. If no comment markers are present the whole distance range of the profile will be used.
If deactivated the inter- and extrapolation is done over the complete distance range of the profile.

start: starts the creation of the Layer-Show-file.

close: terminates the window.

6.3.2 Create LayerShow report

This option allows the creation of a report of the current layershow file. The output is performed either on a printer or on a file with an arbitrary name. The font is selectable under Global (chap. 1.2 - option fonts/ReportFont). You should use a non proportional font like Courier if you want that the colums are correctly displayed. After selecting the option the following input possibilities or choices exist:

filename: allows the input of an arbitrary name or the printer interface, respectively. With specification of LPT1 or LPT2 or COM1 or COM2 or PRN the output is sent to the desired printer interface, for any other specification a corresponding file with the extension REP in the directory ASCII is created.

automatic name: if activated the filename of the report corresponds to the name of the currently loaded profile.

report code: allows the setting of an arbitrary coding. This coding appears in the header of the report.

layers: allows the specification of the layers to output. It is possible to specify individual layers (e.g. 2 or 3) or as well layer sequences (e.g. 2-4) or combinations of these each separated by comma (e.g. 1,3,5-7). The maximum number of layers is limited to 100, as already stated within the layershow creating menu.

start position, end position, increment, average: specify the start and end coordinate in the set dimension, increment the desired distance between 2 subsequent values. Average defines the range over which all values are averaged. By default average is set to increment. If average is chosen larger than the distance between two picks within the layershow file, an averaging is performed over the corresponding number of data points.

thicknesses: activating this option allows the output of the thickness of each layer instead of depth. If an averaging is used for the report output the mean thicknesses are calculated from the individual thicknesses at each location. One exception: with the option “summary comment data” activated the calculation of the mean thicknesses is simply based on the mean depths.

velocities: activating this option allows the output of the current velocities.

amplitudes: The amplitudes are also reported if the option is activated.

core data: if activated the currently loaded coredata are reported at the given positions in addition (see also View MenuItem - option Coredata/1D-models).

statistic summary: If activated a statistic summary is added at the end of the report or at the end of each block defined by a marker comment (option marker comments activated). The summary contains the following information:
1. line: statistics for comment marker
2. line: length of section
3.- n. line: statistic summary for each layer. Each line contains the information for one layer:
average absolute amplitude
average amplitude
average thickness
standard deviation of the average thickness
max. thickness deviation between two adjacent picks
number of cores

max. deviation in % between the core thickness and the thickness of the picks

number of digits: enter the total number of digits for each report value.

decimal places: enter the number of decimal places for each report value.

check amplitudes: activate this option if you want that the amplitudes of the individual picks are controlled before generating the report. If the amplitude of any layer pick is 0 this pick will not be considered and all values for this layer pick within the report output are set to 0. If the amplitudes of all layer picks at a distinct distance are 0 nothing is written out for this position.

xy-coordinates: if activated the xy-coordinates of the receivers stored in each trace header of the current profile are written out within the 2. and 3. colum (see also trace header Edit menu - chap. 2.2). This allows you for example to write out non equidistant xy-coordinates or the xy-coordinates, if the profile is not orientated into one direction (x or y).

start: starts the creation of the layershow report. The report is displayed within a list box. Closing this box terminates the window.

close: terminates the window.

6.3.3 guide for the layershow use

1. pick a distinct reflector - it is not necessary that the reflector is continuous but it also may consist of different broken parts (chap. 6.3.3.1)

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 the layer-show (chap. 6.3.3.2)

6. create a report (chap. 6.3.3.3)

6.3.3.1 pick a distinct reflector

Each reflector must be picked separately. 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:

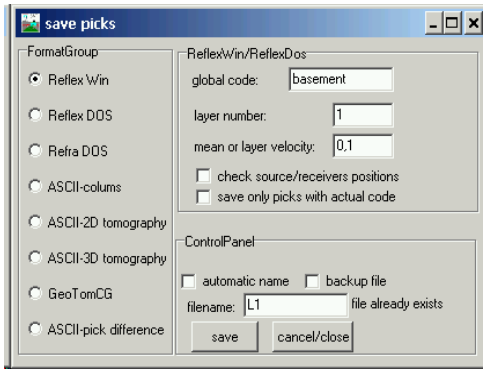
1. **load** the wanted GPR or seismic ZO(zero offset) profile - comment: GPR profiles are normally ZO-lines

2. activate the option **pick**

3. **pick the reflector** using one of the possible picking possibilities:

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
manual pick: use this option for single trace picking or for changing individual picks
auto pick: use the object type continuous reflector

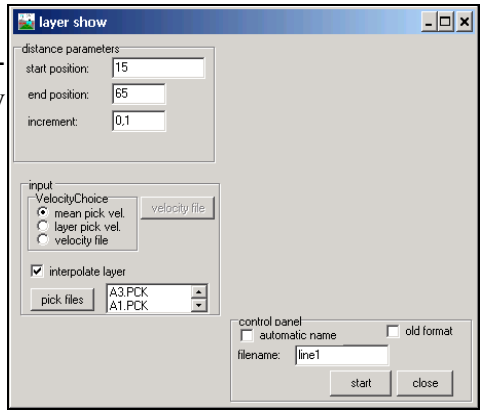
4. use the option **save** for saving the picks of one reflector using the format Reflex Win. Enter the following parameters:
global code: any code
layer number: number of the layer - necessary for the following combination of the different picked reflectors, min. layer number is 0, max. layer number is 100. Each reflector should receive an individual number.
mean or layer velocity: enter a mean or a layer velocity for the depth conversion
filename: filename for the picks (automatic extension is pck)



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

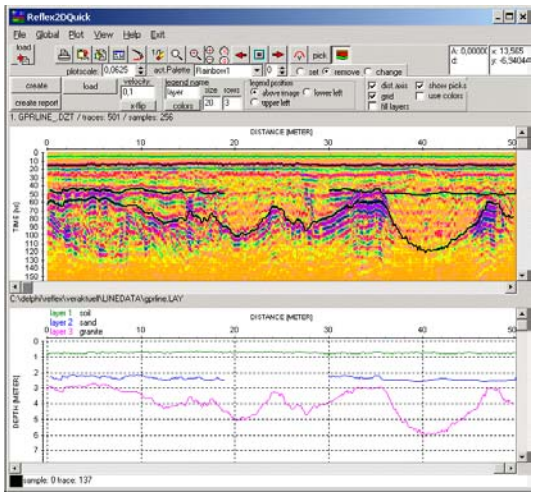
6. **repeat** step 3-5 until all reflectors are picked - to be considered: the layer number must be changed for every different reflector. The max. layer number is 100.

7. activate the option **layer-show**  or under analyse/layer-show - the corresponding ZO-profile must be loaded - the new layershow parameter panel opens.



6.3.3.2 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 100.
The picked traveltimes of the reflectors are transformed into dephts 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 chosen 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.
The option **interpolate layer** controls how the special case of broken reflectors are handled (see below) if layer pick vel. has been chosen.

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

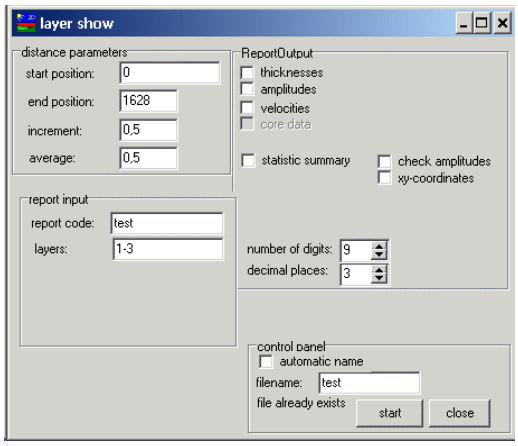
5. activate the option **close** for closing the create layershowy menu.

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 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 **layer pick velocities** 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 current 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 current layer at the point where any of the upper layers is broken.



6.3.3.3 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, currently 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.

7. Export data

Use the option file/export allows to export the current data (including the current processing flow) into different data formats. The following formats are supported: BITMAP (BMP), SEGY, SEG2, GEOTEL, BGREMR, PULSEEKKO, RAMAC- and RAMAC-BOREHOLE, NWF, RADAN and four ASCII-formats (ASCIISYNTH, ASCII-3COLUMS, ASCII-MATRIX(TRACE/ROW) and ASCII-MATRIX(TRACE/COL)).

The option **export format** allows to specify the format of the data to be converted. At the moment the formats BITMAP (BMP), SEGY, SEG2, BGREMR, GEOTEL, PULSEEKKO, NWF, RAMAC, RAMAC-BOREHOLE, RADAN and five ASCII-formats (ASCII-3COLUMS, ASCII-4COLUMS, ASCII-SYNTH, ASCII-MATRIX(TRACE/ROW) and ASCII-MATRIX(TRACE/COL)) are supported.
ASCII-3COLUMS: each line contains the distance, time and the amplitude (format 12:3). Depending on the settings within the AsciiColums box the distance is either determined from the fileheader (**fileheader coordinates** activated) or set to the distance stored within the trace header (**traceheader coordinates** activated). Example for the use of the option traceheader coordinates: the data are nonequidistant and you want to write out the correct positions.
ASCII-4COLUMS: each line contains the x-coordinate, y-coordinate, time and the amplitude (format 12:3). Depending on the settings within the AsciiColums box the x- and y-coordinates are either determined from the fileheader coordinates **(fileheader coordinates** activated) or they are set to the CMP-coordinates stored within the trace header (**traceheader coordinates** activated). Example for the use of the option traceheader coordinates: the data are nonequidistant and you want to write out the correct CMP-positions.
ASCII-MATRIX(TRACE/ROW): each line of the ASCII-file contains the amplitude values of one trace starting at trace 1 (row 1 = trace 1, row 2 = trace 2, ...). No header information is written out.
ASCII-MATRIX(TRACE/COL): each column of the ASCII-file contains the amplitude values of one trace starting at trace 1 (column 1 = trace 1, column 2 = trace 2, ...). No header information is written out.
ASCII-GEOSOFT_XYZ: each line contains the distance, time and the amplitude (format 12:3). Depending on the settings within the AsciiColums box the distance is either determined from the fileheader (**fileheader coordinates** activated) or set to the distance stored within the trace header (**traceheader coordinates** activated). Each scan in x-direction is separated from the previous one by a command line: "line xxx". The parameters **XSCALE** and **YSCALE** define a scaling value for the x-values and the y-values respectively. The parameter **threshold** defines a threshold value. Data values (absolute value) below this threshold are not written out.
ASCII-GRD: this format corresponds to the ASCII-GRD format used by the program SURFER. The radiobox **first sorting** defines how the data are written out onto the ASCII-file. With **y-(time)-direction** activated the data are written out trace after trace. Then the second line contains the number of samples and of traces. The third line contains the start and end-position in time-direction, the fourth line the start and end-position in profile direction. Each of the following lines contains all amplitude values of one trace. With **x-(distance)-direction** activated the data are written out sample after sample. Then the second line contains the number of traces and of samples. The third line contains the start and end-position in profile-direction, the fourth line the start and end-position in time direction. Each of the following lines contains all amplitude values of one sample.
To be considered for timeslices: in order to get the correct sorting within Surfer it might be necessary to use the x(distance)-direction for the first sorting.
ASCII-3D_DATA: this format should be used for 3D-data. Each line contains the x-, y-coordinates and the time and the amplitude. An exponential format is used. With the option depths activated the converted depths based on the given velocity are written out in addition. Then each line contains the x-, y-coordinates, the time, the converted depth and the amplitude.
SEGY: With the option **ps timeincr.** activated the segy-timeincrement within the exported data is

assumend to be in picosec if the original timeincrement is in ns. The option should be activated for ns-data because SEGY only allows to store the timeincrement as an integer value. If the option is deactivated the program stores the timeincrement in µs which will be 0 in most ns-timeincrement cases because of this integer restriction.
The parameter **scaling factor** is used for scaling the distances before export. The option is only necessary for the export format SEGY. The following bytes are written out on the traceheaders:
001-004: trace sequence number
009-012: original field record number
029-030: trace id code (1 = seismic)
037-040: distance from source to receiver
071-072: scaler for elevations and depths
073-076: Source x-coordinate
077-080: Source y-coordinate
081-084: group x-coordinate
085-088: group y-coordinate
089-090: coordinate units (1=length)
115-116: Number of samples in this trace
117-118: sample interval in µs
159-160: day of year
161-162: hour of day
163-164: minute of hour
165-166: second of minute

BITMAP: The following bitmap formats are supported: bmp, jpg, tif and pcx. The parameters **AutoSize**, **XSize** and **YSize** determine the size of the bitmap. With AutoSize activated the x- and y-sizes of the bitmap are automatically determined from the number of scans and samples. With AutoSize not activated you must manually enter the x- and y-sizes. If the entered x- or y-size number exceeds the trace or samplenumber an automatic interpolation in the corresponding direction is done for the pointmode presupposed the option **autointerpolation** is active. The max. x- or y-size in pixels is 30000. No flipping and rotation of the axis (see plotoptions) are supported.
If the depthaxis under Plotsuboptions (chap. 1.7.6) is activated or if the y-axis already is a depth axis the option **scale relationship** allows you to enter a scale for the depth and therefore also the timeaxis. In this case the y-size is determined from the x-size and this scale value. For example a number of 1 for the scale means that the x- and the depthaxis have the same scale. Example: if the x-axis of a file consisting of 2000 traces is 100 m long and the depth-axis is 10 m long (depth-axis/x-axis = 1/10), the bmp-file will have about 2000 x 200 pixels if the scale relation is set to 1. If the scale relation is set to 2, the number of pixels for the depth-axis will double and the bmp-file will have 2000 x 400 pixels, approximately. The option is available both with activated and deactivated AutoSize option.
If axis within the PlotOptions is activated, the axis in x- and y-direction are automatically added to the bitmap using the current font settings.
With the option **show name** activated the filename (without path) is shown at the top using the textfont.
With the option **showpicks** activated it is possible to display picks stored within individual pickfiles. After having pressed button start you must choose the wanted pickfiles from the filelist. Within the fileheader of each pickfile the corresponding datafilename is stored (see also Pick save MenuItem). This filename serves for the assignment of the chosen pickfiles to the chosen 2D-datafiles.

Start starts the export. The converted files will have the same name as the original file except the extension and will be stored under the path ASCII under the project directory.

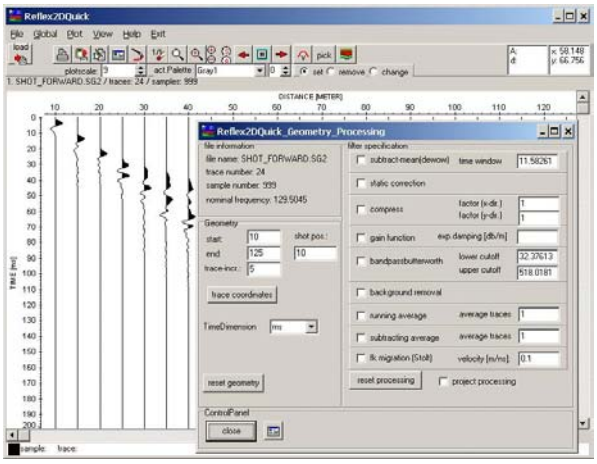
8. Application guide

8.1 seismic refraction single shot data

The program Reflex2DQuick is well suited for the picking of first arrivals for a subsequent traveltime interpretation using for example the Reflexw traveltime analysis module or any other program which allows to import ASCII traveltimes.
Two seismic shot example data are automatcially installed: shot_forward.sg2 and shot_reverse.sg2. In the following the procedure for such a file is described:

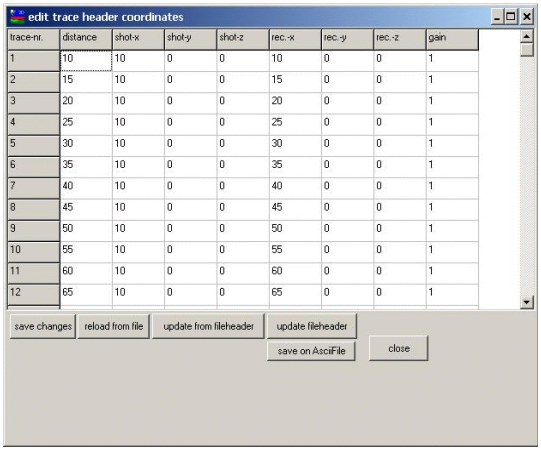
1. Enter the program **Reflex2DQuick**

2. Activate the option **load** and load the wanted datafile with the correct filetype (this example: filetype: SEG2).



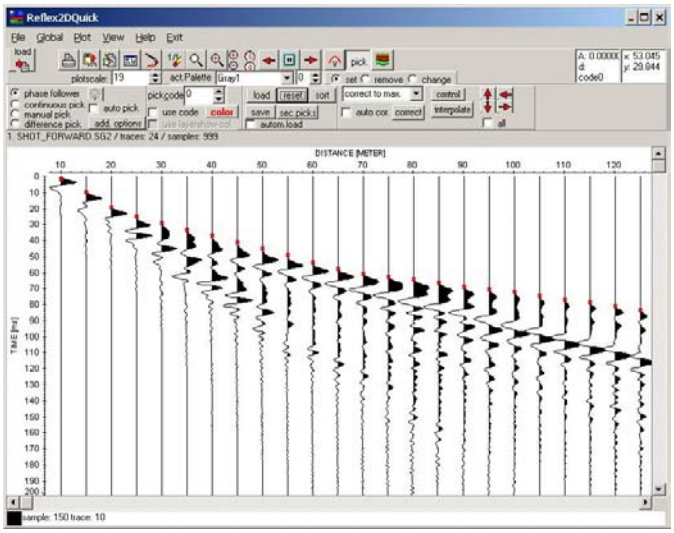
3. The **Reflex2DQuick_Geometry_Processing** menu (see also chap. 2) opens together with the data. Now you must specify the geometry as well as the filter (if necessary) and the plotoptions.

4. set the **plotoptions** (see also chap.3): normally the wiggle mode together with activated option tracenormalize (plotscale depends on the number of receivers, e.g. 20 for 24 receivers) is the best choice for single shot refraction data:
Plotmode: Wigglemode
tracenormalize activated
XYScaledPlot activated

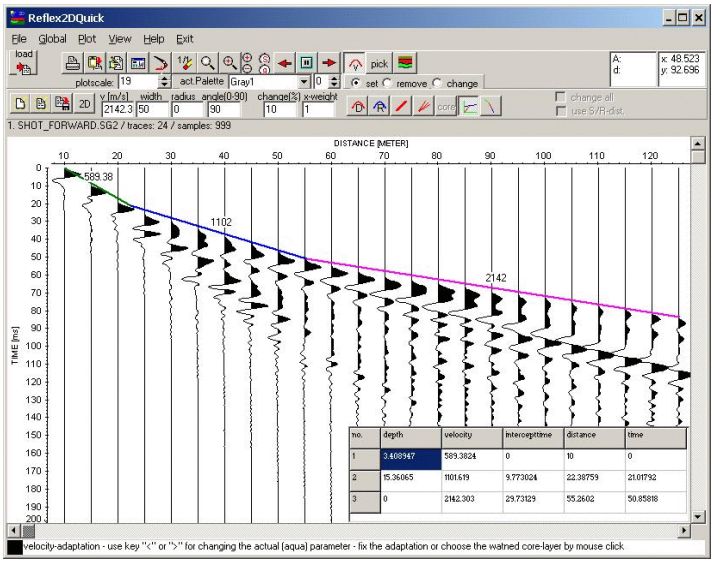


5. set the **geometry** (see also chap. 2): often the original SEGY or SEG2 data already contain the informations about the receiver and shot positions. These informations can be viewed and changed using the option trace coordinates. Then the edit traceheader coordinates menu opens (see picture on the right and chap. 2.2.1). To be considered for manual editing: distance value and rec.-x value should be equal for each trace. Use the option save changes in order to update the traceheadercoordinates.
The overall start and end of the receiver line and the shot position are also given by the parameters
start: start of the receiver line
end: end of the receiver line
shot-pos.: position of the shot
within the Reflex2DQuick_Geometry_Processing menu. These parameters can be used for a simple definition of the reciever line if not already defined within the original data. For that purpose please enter the start and end position and the shot position, then activate the option trace coordinate and then activate the option update from fileheader within the now open edit traceheader coordinates menu.

6. Set any **filter** if necessary and close the window.



7. The data are now ready for **picking** the traveltimes (see also chap. 6). It is recommended to use the automatic name for saving the picks. It is not necessary to enter the layer number and the velocity within the save picks menu. These parameters are only necessary for reflection data. The warning message may be ignored. If the picked traveltimes shall be interpreted by an external program any of the ASCII-formats should be used. For a later use with the Reflexw tomography either the ASCII-2D or ASCII-3D tomography formats can be used. With the option "export several existing picks into 1 ASCII-file" activated you may export several existing pickfiles into 1 ASCII-file.



8. Do this procedure for all wanted shots. Then the picked traveltimes are ready for interpretation.

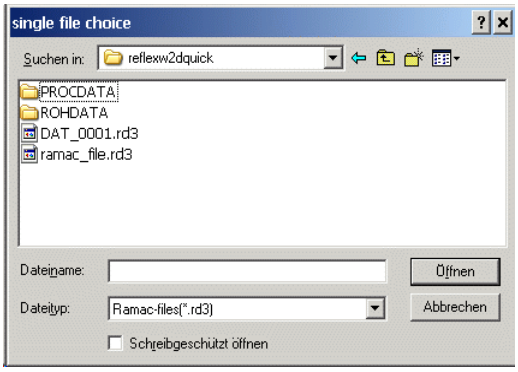
9. You also may perform a simple **intercepttime interpretation**. For that purpose load the wanted shot, enter the option analyse/velocity adaptation and activate the option intercept time analysis. Now click on the data and move the cursor with clicked mouse button to the first bending point and leave the mouse button. The first point is automatically set to time zero and to the shot position. Activate again the left mouse button and move to the next bending point, and so on. The velocities derived from these settings must increase with depth. After having finished the settings the depth and velocities of the calculated 1D-model are shown in a window.

To be considered: if the geophone positions are not equidistant you must activate the plotoption traceheader distances in order to plot the traces at these positions stored within the traceheader distance positions (distance position should be equal rec.x position).

8.2 GPR constant (zero) offset data

The program Reflex2DQuick allows a fast access to 2-dimensional constant (zero) offset GPR or seismic (ultrasound) data. Some GPR zero offset data are automatcially installed: dat_0001 and ramac_file are two files originating from the Mala GPR-system, gprline_.dzt has been acquired using a GSSI GPR-system and the files xp*.* are 6 parallel GPR-files. In the following the procedure for the file dat_0001 is described:

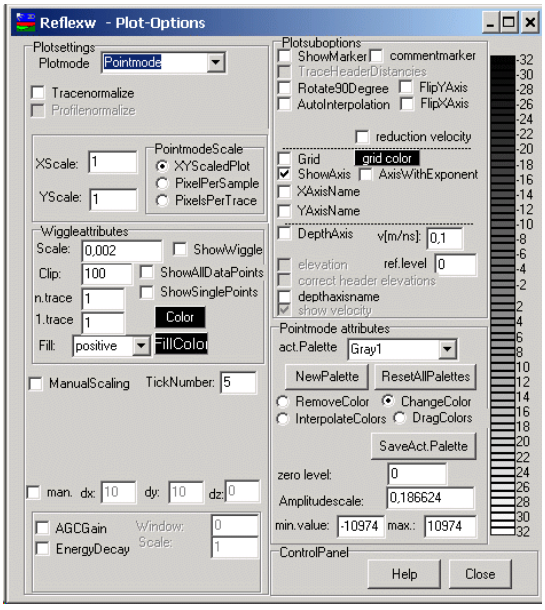
1. Enter the program **Reflex2DQuick**



2. Activate the option **load** and load the wanted datafile with the correct filetype (this example: filetype: rd3).

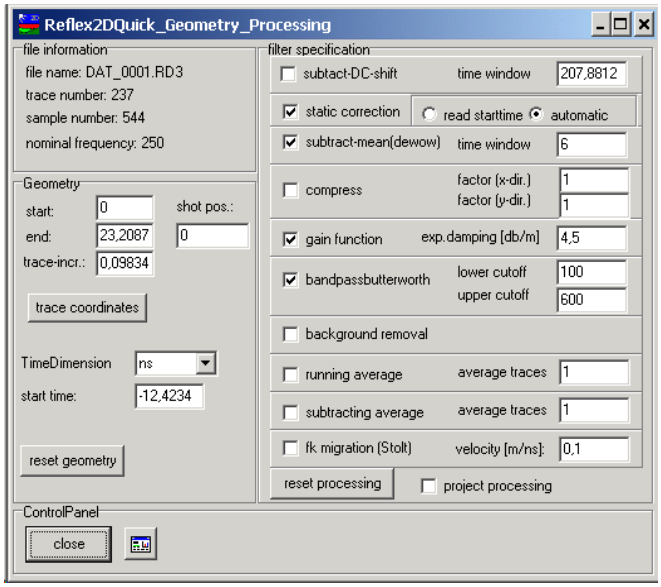
3. The **Reflex2DQuick_Geometry_Processing** menu (see also chap. 2) opens together with the data. Now you must specify the geometry (if necessary) as well as the filter and the plotoptions.

4. set the **plotoptions** (see also chap.3): normally the point mode together with deactivated option tracenormalize (plotscale depends ont the amplitude values) is the best choice for GPR zero offset data:
Plotmode: pointmode
tracenormalize deactivated
XYScaledPlot activated



5. set the **geometry** (see also chap. 2): often the original data already contain the informations about the traceincrement (parameter trace-incr.). The start and end of the profileline are given by the parameters start and end. The parameter shot pos. is of no interest. It will only be used for single shot data (see chap. 8.1). The timedimension is set to ns for all GPR formats and to ms for SEGY or SEG2 data. If the SEGY or SEG2 data are GPR data the timedimension must be changed manually.

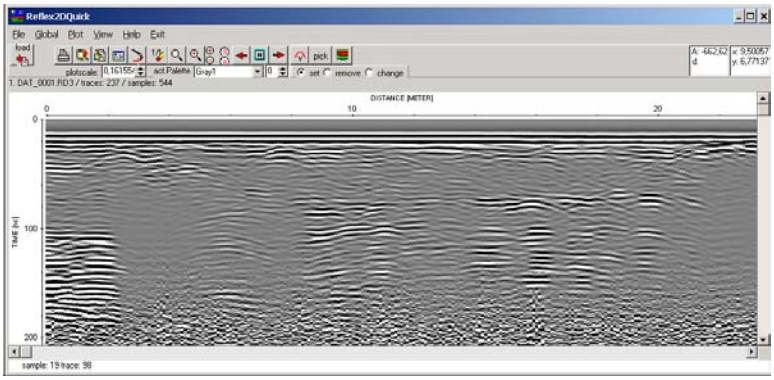
6. Set the necessary **filter**. A useful filtersequence for Mala based GPR-data is given within the example on the left. If several datafiles have been acquired using the same field parameters and under the same conditions it might be useful to use the project processing option. Activating this option allows you to load previously defined filter and geometry settings which are valid for all subsequently loaded datafiles.

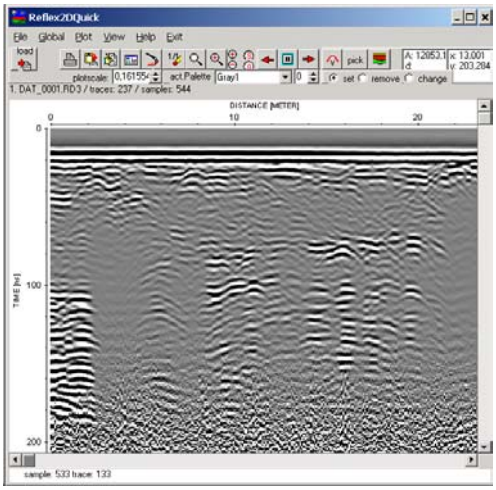


7. Close the menu and the data will be replotted based on the given settings. They are now ready for the **interpretation** like the velocity analysis (see also chap. 5) or picking the onsets and create a layer model (see also chap. 6).

Some remarks:

• The discernibility of the indiviudual elements strongly depends on the scale of the distance and time axis. The following pictures show an example of the same file with different scales using the PointModeScale XYScaled. The small scale diffraction elements are much more clearer in the right window.





• The program can be started twice. Thereby it is possible to compare two different files or one file with different filter specifications.