

Picking different reflectors within REFLEXW

In the following the picking of different reflectors within REFLEXW is described including the different picking possibilities, defining velocities for the depth conversion and the combination of different reflectors into a so called layer show picture.

I. General flow

1. pick a distinct reflector - it is not necessary that the reflector is continuous but it also may consist of different broken parts (chap. II)
2. save the picks of one reflector using a special name (e.g. layer 1)- enter the layer number, the mean velocity and a layer code
3. repeat step 1 and 2 until all reflectors have been picked
4. enter the layer-show
5. create a velocity file if layer velocities shall be used instead of mean velocities (chap. III)
6. create the layer-show (chap. IV)
7. create a report (chap. V)

II. 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. enter the module **2D-dataanalysis**
2. **load** the wanted GPR or seismic ZO(zero offset) profile - comment: GPR profiles are normally ZO-lines
3. activate the option **pick**
4. **pick the reflector** using one of the possible picking possibilities:

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 (from ver. 2.5)

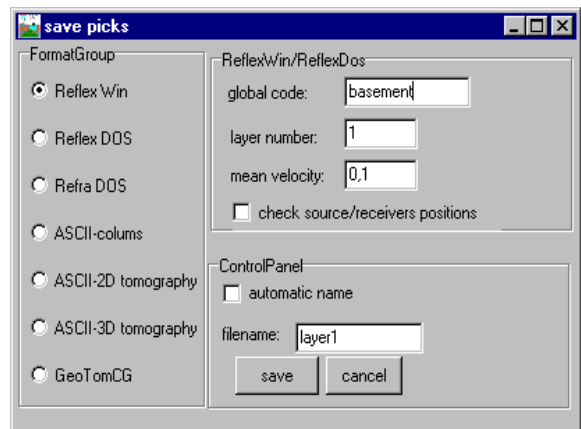
5. use the option **save** for saving the picks of one reflector using the format Reflex Win. Enter the following parameters:

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 10 (up to ver. 2.1) or 100 (from ver. 2.5). Each reflector should receive an individual number.


mean velocity: enter a mean velocity for the depth conversion

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



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

7. **repeat** step 4-6 until all reflectors are picked - to be considered: the layer number must be changed for every different reflector. The max. layer number is 10 until ver. 2.1.2 (100 from ver. 2.5).

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

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

For the **time-depth conversion** using the option create LayerShow (see section IV) it is necessary to take a **velocity** into account.

The first possibility is to use the **constant mean velocities** for the total overburden of every pick file, which were set during the pick process (see section II - step 5). If you want to use this option you may skip this chapter and you may continue with chapter IV.

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

comment line

comment line

1.velocity line

....

n. velocity line

Each velocity line contains:

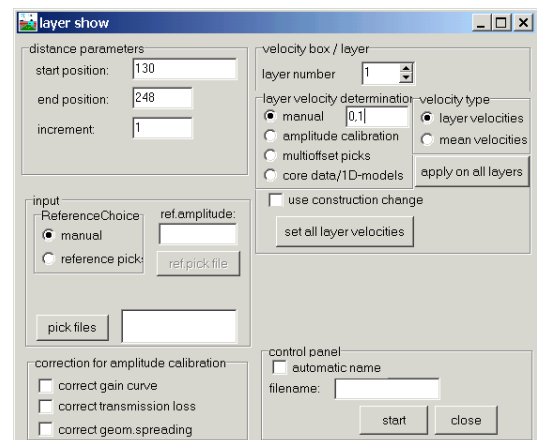
distance v-layer1 v-layer2 v-layer10 or v-layer 100 (from ver. 2.5)

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

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

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

2. there are four different ways to generate an ASCII velocity datafile: The layer velocities are set **manually**, which implies no lateral change of the velocity in one layer, or the velocities are calculated using an **amplitude calibration** of the picked layer boundaries, or the velocities are calculated based on different **multioffset picks** or they are taken from **core data**. The second case (amplitude calibration) is only integrated within REFLEXW for completeness. We do not recommend this method (see manual, chap. 1.12.3.4). You may use different ways for the different layers.



3. Activate the option manual and enter the **velocities** for the individual layers and the **filename** for the velocity file (the velocity file automatically has the extension vla).

4. activate the option **start** for the generation of the ASCII file. The file is shown in a window and activate the option close for closing the display of the file.

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

6. The ASCII-file can be changed manually within any editor in order to take into account lateral velocity variations.

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

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

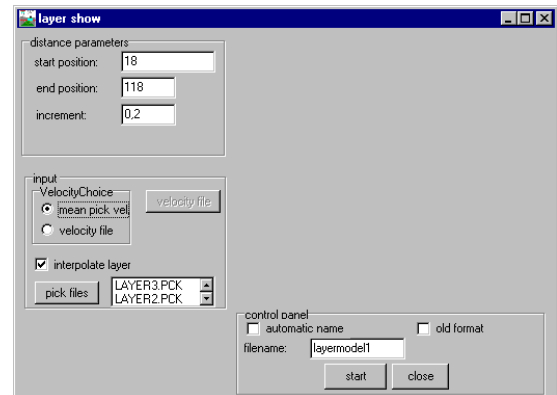
The picked traveltimes of the reflectors are transformed into depths using either a mean velocity for a distinct layer or based on a laterally varying velocity file.

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

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

2. activate the option **pick files** within the layer show menu and choose all wanted pickfiles (multiple choice using the shift or str-key) - the 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.



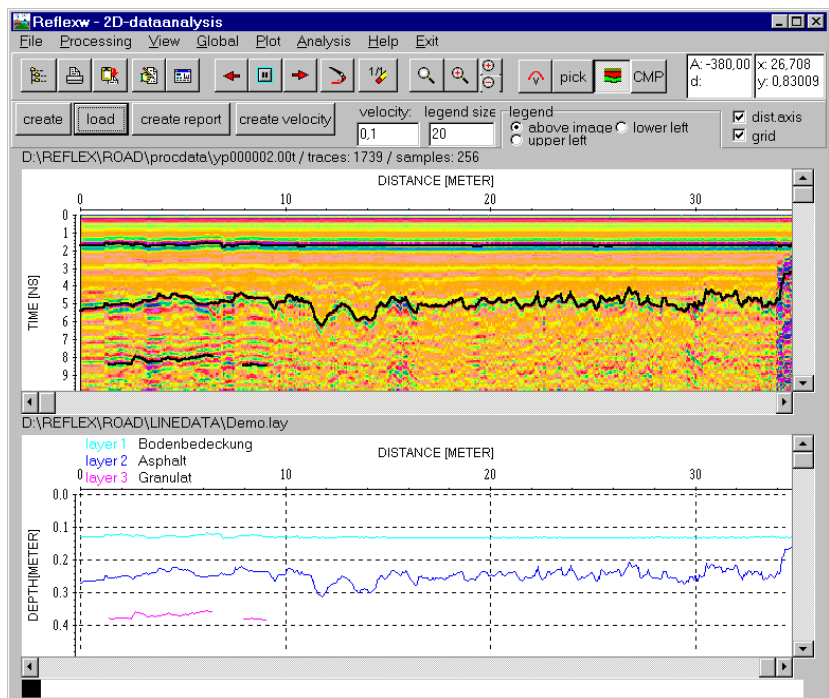
If you want to use a **velocity file** for the time-depth conversion instead of the **mean velocity** stored with each pick file first you must create such a velocity file (see section III). Then you must activate the option velocity file and load the wanted velocity file. The option **interpolate layer**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

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

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

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

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

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

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

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

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

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

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

The screenshot shows the 'layer show' dialog box with the following settings:

- distance parameters:** start position: 18, end position: 118, increment: 0,2, average: 0,2
- report input:** report code: test, layers: 1-5
- ReportOutput:**
 - thicknesses
 - amplitudes
 - velocities
 - core data
 - marker comments
 - comments
 - amplitude-norm.: low, mean, high, unnorm.
 - check amplitudes
 - report only construction data
 - summary construction data
 - reverse chainage
 - number of digits: 9
 - decimal places: 3
- correction:**
 - correct gain curve
 - correct transmission loss
 - correct geom. spreading
- control panel:**
 - automatic name
 - filename: report1
 - start, close buttons