Introduction to the interpretation of seismic refraction data within REFLEXW

In the following the complete interpretation of seismic refraction data is described including import of the seismic data, picking the first onsets, putting together the picked traveltimes, assigning to specific layers, doing the layer inversion and refining the resulting model by raytracing (chapter I to chapter III). Another possibility of interpreting seismic refraction data is the refraction-tomography, which is presented in chapter IV.

Furthermore is shown in chapter V, how the results of these two independent methods are used to get reliable information about the investigated area.

As the interpretation of seismic refraction data measured along a line with topography is widely done in the same manner as the interpretation of seismic refraction data along a line without any topography, we firstly explain the interpretation of seismic refraction data in general (chapter II), before going in details, how the topography can be taken into account (chapter VI).

I. Import the data and pick the first onsets (done within the module 2D-data-analysis)
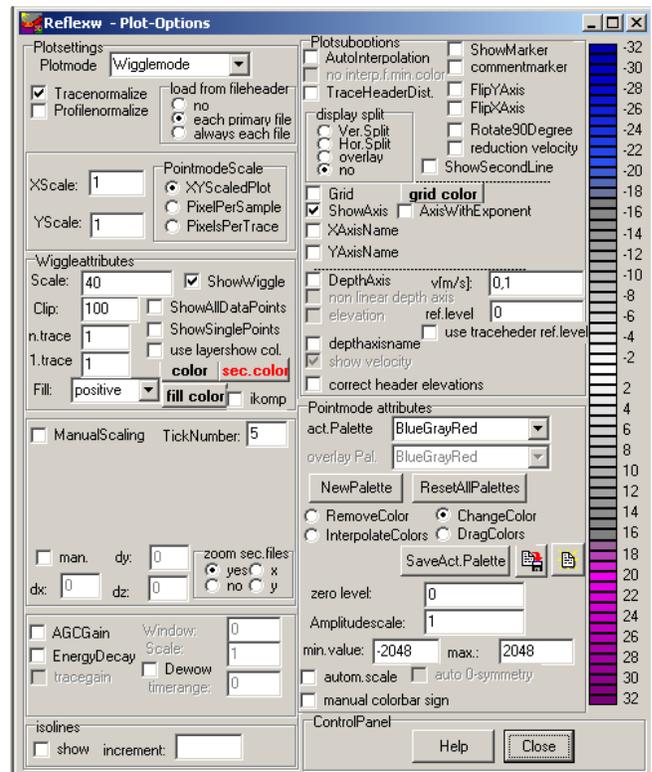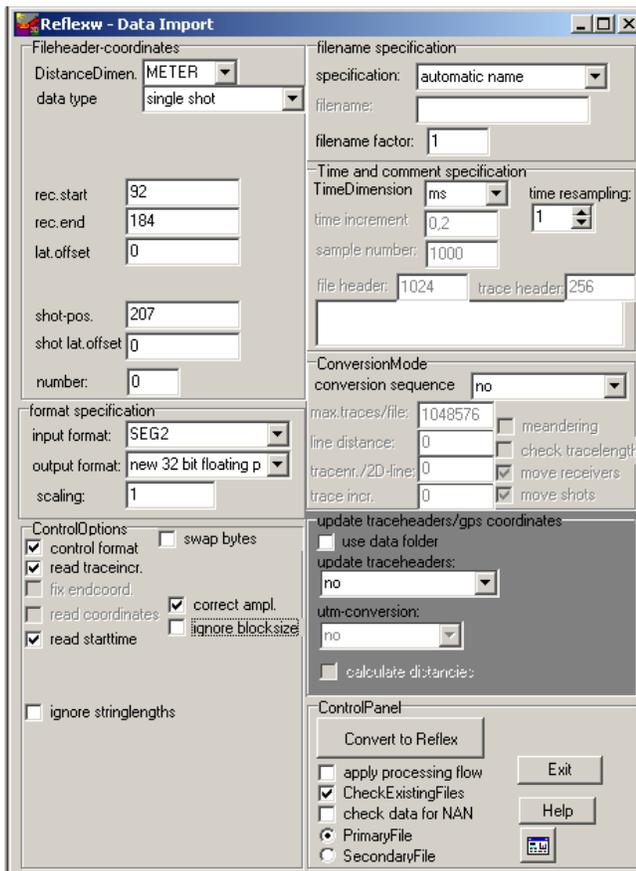
1. Enter the module **2D-dataanalysis**.

2. Activate the option **import**.

3. Choose the following options within the **import menu** (left figure above):

data type: single shot
rec.start: start of the receiver line
rec.end: end of the receiver line
shot-pos.: position of the shot

outputformat: new 32 bit floating point for a higher data resolution

To be considered for SEGY or SEG2-data: The option **swap bytes** controls if the original data originate from UNIX (activate option) or DOS-machines (deactivate option). If the conversion fails try to change this parameter.

The following **plot option**s (right figure above) should be set (the option may be activated within the import menu (the speed button below the help option):

Plotmode: Wigglemode
tracenormalize activated
XYScaledPlot activated

4. Activate the option **Convert to Reflex** and choose the wanted original datafile from the filelist.

REFLEXW uses the individual **traceheader coordinates** stored with each trace for the further traveltime analysis. The coordinates defined above (shot-pos. and rec.start and rec.end) only serve as so called header coordinates and may be used to actualize the traceheader coordinates (see below) if these are not correctly defined within the original data. REFLEXW automatically reads in the shot and receiver positions of the individual traces if these are defined within the original files and stores them into the traceheader coordinates. After the import the traceheader coordinates are automatically shown within a table.

If the traceheader coordinates are not correct (e.g. because they are not stored within the original files) these coordinates must be defined separately:

a. within the table use the option **update from fileheader** inside the UpdateGroupBox and then **save changes** in order to actualize the traceheadercoordinates based on the entered fileheader coordinates (shot position, rec. start and end, see above).

b. within the **traceheader menu**. The traceheader menu is entered within the **fileheader menu** using the option **ShowTraceHeader**. Here different actualization options are available. You may choose either fileheader, ASCII-file or table and then you must press the button **update** to actualize the traceheader coordinates. For a further description of the individual options please refer to the online-help of the traceheader menu.

If the traceheader coordinates are read in correctly from the original data you also may use the option **update fileheader** in order to actualize the fileheader coordinates from the traceheader coordinates. In this case there is no need to enter the correct rec./start and end coordinates and the shot position within the import menu.
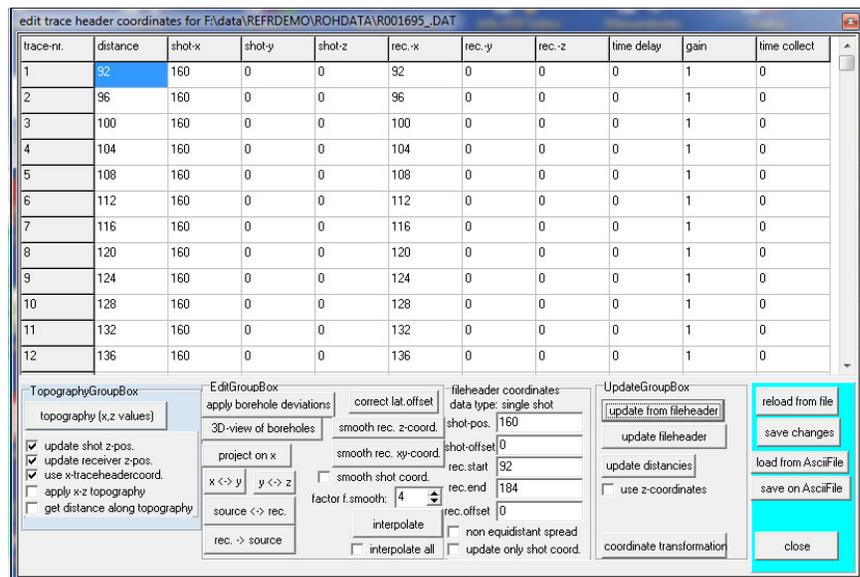
5. Do any **processing** if necessary or change the settings within the **fileheader** (option edit/fileheader), e.g. the start time, ... Note: Filtering the data can lead to wrong first arrivals!

6. **Pick** the first arrivals and save them on file.
- Use the PlotoptionsMenu if necessary to adjust the display of the data to ease the identification of the first arrivals. (E.g., set scale to 70, and clip to 100 and fill to negative.)
- Click on pick and pick the <u>very first arrivals</u> (see left figure below).
  For traces close to the shot point it might be different to pick the correct first arrival. In such cases it might be better to skip these traces instead of picking wrong events. The zero traveltimes can be inserted later on in the module **traveltime analysis**, see below.
- Save the picks using the option save. 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 (see right figure below).

7. Do this procedure for all wanted shots. Then the picked traveltimes are ready for interpretation.
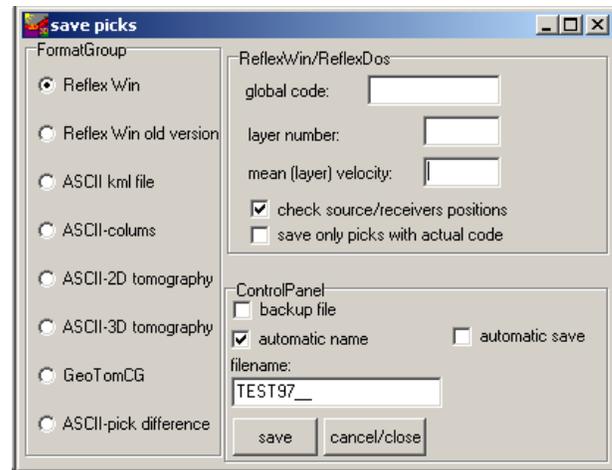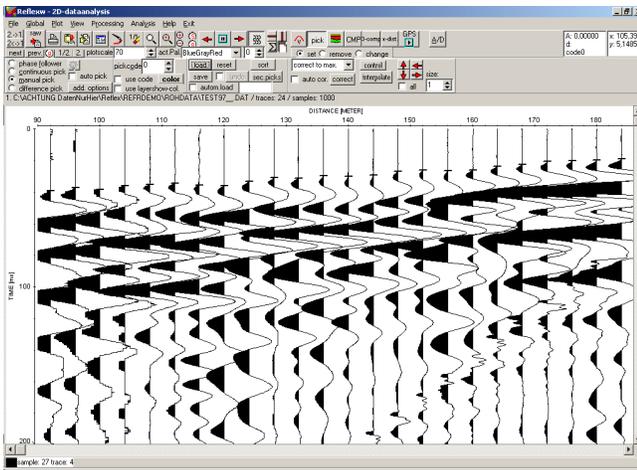
8. You also may perform for every shot a simple intercepttime interpretation within the 2D-dataanalysis module.
For that purpose load the wanted shot, enter the option analyse/velocity adaptation and activate the option intercept time analysis. Now click on the data and move the cursor with clicked mouse button to the first bending point and leave the mouse button. Do not simply click on the bending points because this yields wrong velocities. The first point is automatically set to time zero and to the shot position. After having released the mouse button at the first bending point activate again the left mouse button and move to the next bending point with pressed mouse button, 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).
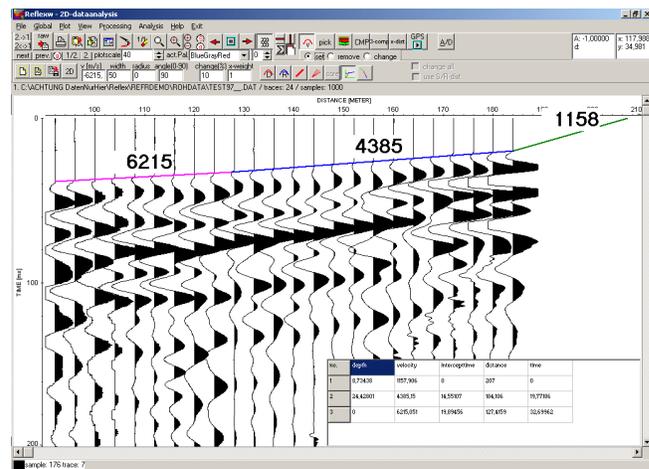
II. Interpretation of the picked traveltimes (done within the module traveltime analysis)

1. Enter the module **traveltime analysis**.

2. **Load** the wanted traveltimes which shall be interpreted together - option file/load traveltimes - multiple choice using the shift- or ctrl-key.
In order to display the data in 'refraction mode' (i.e. the time axis faces upwards) please activate the plot option **FlipYAxis**. If the option **colored** is activated, every shot is displayed in a certain color. This may help to get a first overview of the picked traveltimes.

3. Use the option **edit/InsertshotZerotraveltime** in order to insert a zero traveltime at all shot points if this zero traveltime is not still defined. This is useful in order to get a better determination of the velocities of the uppermost layer.

4. Enter a filename for the actually put together traveltimes and save them on disk.

5. Increase the **layer-no** to 1.

6. Activate the option **assign** in order to assign the wanted traveltimes to layer 1. (It is recommended to deactivate the option **colored** now.)

7. Assign the traveltimes to layer 1 by using the different possibilities. Assign all traveltimes of one shot until a distinct change of the apparent velocity within the traveltime curve occurs. This change is sometimes not easy to determine. Use the options **Forward**, **Reverse** and **keep last shot** for a better definition at those dataparts, where the pick position is not well defined (e.g. overlapping files) - for further information refer to the online help. The traveltimes assigned to layer 1 will be highlighted (default color green), save the traveltimes. For shots outside the receiver line do not assign those traveltimes which have only been inserted and do not have at least one additional real datapoint (within this example shot 12 and 13) because then a wrong velocity would result.

8. Activate the option **combine** in order to do the inversion for the first layer. If no topography is given, do not activate the topography in the CombinePanelLayer1. If topography is given, please refer to chapter VI.
(To be considered: The inversion for the first layer only consists of the determination of the velocity.)

9. Activate the option **wavefront-inversion** and a new window (the modelling window) opens and a
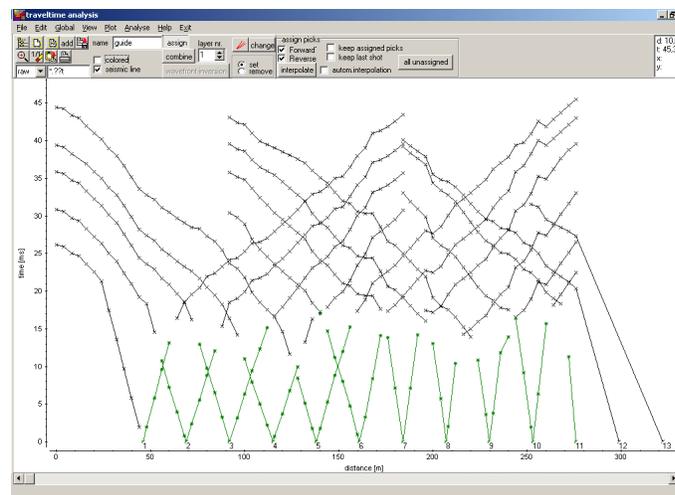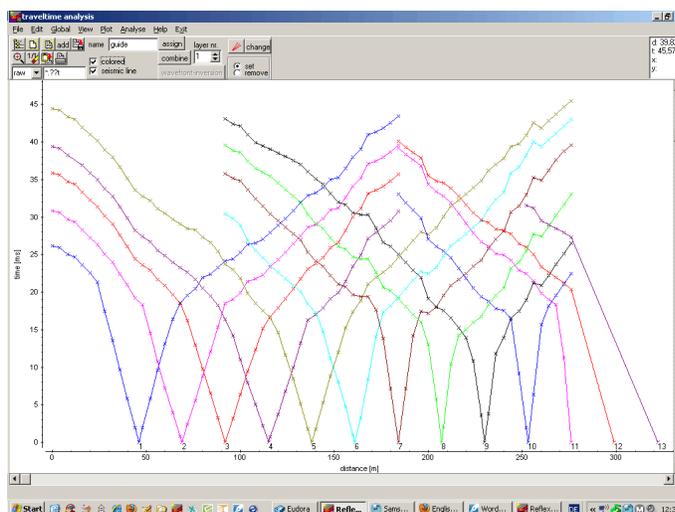
model has been automatically created consisting of one layer with the velocities taken from the linear regression analysis of the assigned traveltimes.

10. Do some changes of the model if you want (e.g. remove some unwanted velocity points, adapt the model size) - to be considered: the option topography should be deactivated. Increase zmax in such a way that all layers to be inverted fall into this depth range.

11. Enter a modelfile name and save this model on disk.

12. Close the modelling window.

13. Increase the **layer-no**. to 2.

14. Activate the option **assign** in order to assign the wanted traveltimes to layer 2.

15. As for layer 1 assign the traveltimes to layer 2 by using the different possibilities. The travel-times assigned to layer 2 will be highlighted (default color blue). Save the traveltimes.

16. Activate the option **combine** in order to do the inversion for the second layer.

17. Enter the **forward** and the **reverse shot numbers** - these numbers define the range for doing the inversion of the actual layer.

18. Activate the option **generate** when the option autocombine is chosen - a complete combined traveltimecurve (forward and reverse) is generated and the total forward and reverse traveltimes are shown.

19. If the total forward and reverse traveltimes differ significantly (e.g. more than 5 ms) activate the option **balance** in order to balance the forward and reverse traveltime branches. A large difference can be have difference causes:
- wrong assignment of the picks
- too large gaps between the picks (an interpolation should be avoided if possible)
- 3D-effects
In any case the inversion is more accurate the smaller the difference between forward and reverse traveltimes. If the difference is too big it might be useful to perform the inversion for several partions.
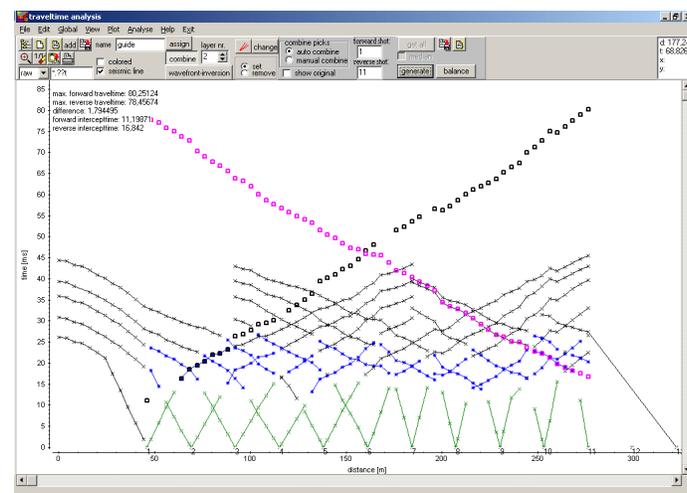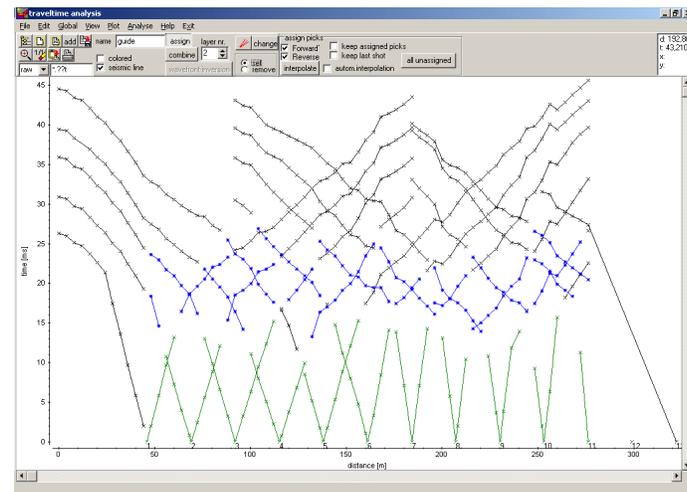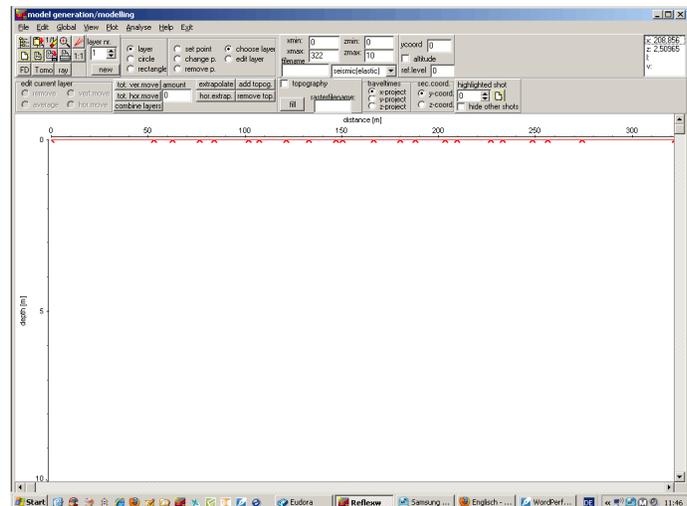
20. Activate the option **wavefront-inversion** - a file choice window opens and you must choose the modelfile containing the first layer already interpreted.

21. The chosen modelfile is shown within the modelling window – the inversion must be started manually within the **RayGroupBox** on the right hand side of the model. To be considered: The max. depth of the model must be chosen in such a way that the max. estimated depth of the layerboundary to be inverted is smaller than this max. depth value.

22. Enter the wanted increment **DeltaX** for the wavefront-inversion (e.g. 0.5 m) and the expected number of different velocities for the new refractor.

23. Start the wavefront-inversion using the option **start**. Note: If the inversion fails (because of not setting an appropriate max. depth values, e.g.) do not use the options start to start the inversion a second time. Instead doing that, please shut the modelling window and activate again the option wavefront-inversion inside the module traveltime analysis 2D to start the inversion again (ref. to 20).

24. At the end of the inversion the velocity determination menu for layer 2 velocity appears. If the entered number of different velocities (see item 22) is greater 1 the traveltime branches are automatically subdivided into a number of different linear regression curves of which you may interactively change the start and end position by simply clicking on it and drawing with pressed left mouse button. If its o.k., close it. The inversion is finished and the new layer boundary is plotted into the model. If more than one velocity for the refractor had been chosen it is possible to interactively choose the different lines of best fit.
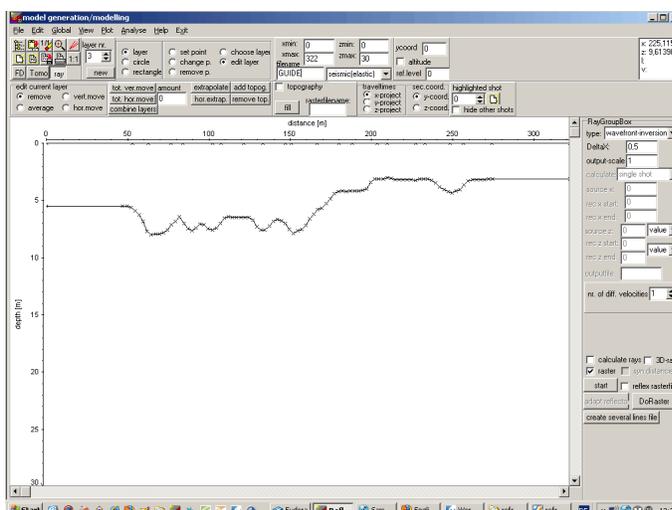
25. Extrapolate the inverted boundary to the model borders (option hor.extrap. or extrapolate) and do some changes of the model if you want to - for example: smoothing is often useful and sometimes some artefacts at the outer borders occur which must be eliminated.

26. Save the model using the old or another filename.

27. Close the modelling menu.

28. Repeat step 13 - 27 for the next layers. If all remaining traveltimes belong to the last layer (e.g. layer 3), you can use the option **all unassigned** to assign them.

29. After having done the complete inversion the inverted model should always be checked (and changed if necessary) using the forward raytracing included within the modelling menu, refer to chapter III.
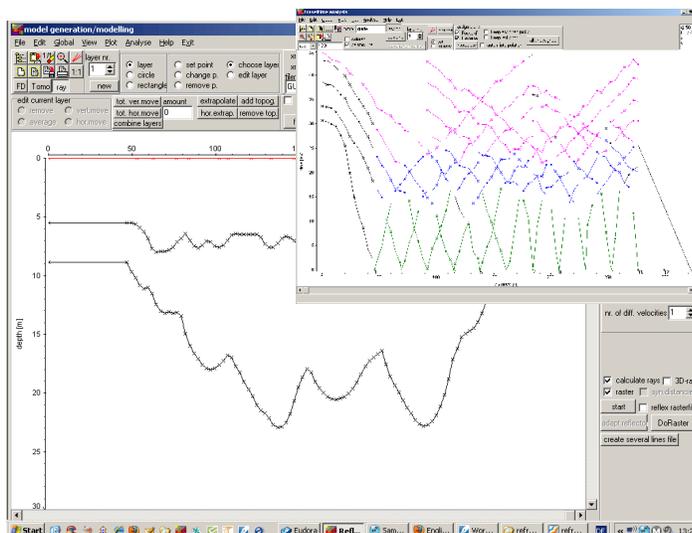
III. Forward raytracing to refine the model (done within the module model generation/ modelling)

The ray tracing modelling tool allows the traveltime simulation of seismic wave propagation by means of a finite difference approximation of the eikonal equation. The calculation of the synthetic traveltimes is restricted to the first arrivals and reflections for an arbitrary 2-dimensional medium. No secondary arrivals can be calculated.
 The raytracing may be used for
- the control of an inverted model and
- an iterative adaptation of the calculated and real data by stepwise changing the underground model.

1. Enter the module **model generation/ modelling**.

2. **Load** the inverted model using the option file/load model.

3. Activate the option **ray**.

4. The **Ray-GroupBox** opens in addition (see figure on the right). Within this group box you have to enter the necessary raytracing parameters, see below.

5. As you want to simulate the observed traveltimes of different shots along the line, you have to load the observed traveltimes using the option File/**load data traveltimes**. Then the screen is split vertically showing in the upper window the model and in the lower the data.
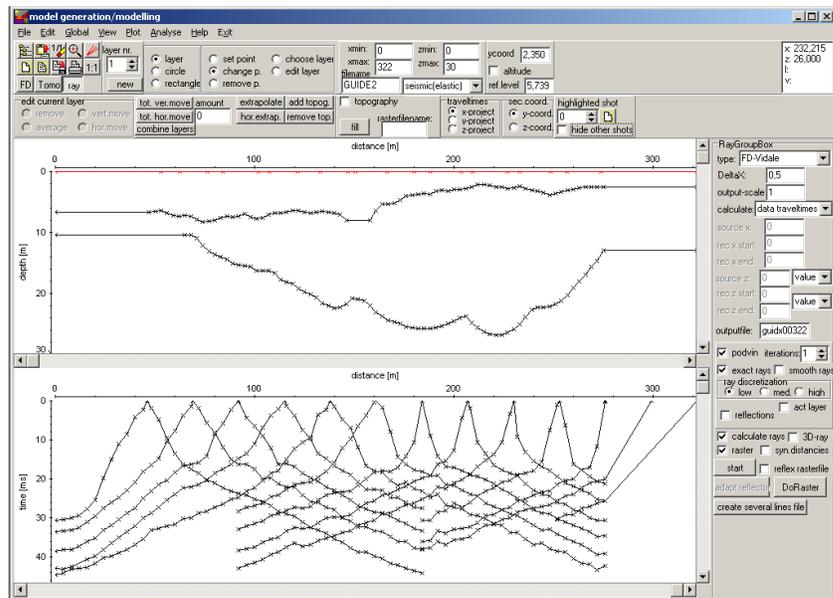
6. Now the ray tracing parameters have to be chosen:
- enter the wanted raytracing **type** FD-Vidale.
- enter the gridding increment **DeltaX** (equal in x- and z-direction - should be in the range of the receiver increment or less - depends on the model complexity), e.g. 0,5.
- enter the **output-scale**, e.g. 1.
- enter the **calculate** type - in this case **data traveltimes** because we want to simulate all loaded observed traveltimes.
- enter the **outputfile** name.
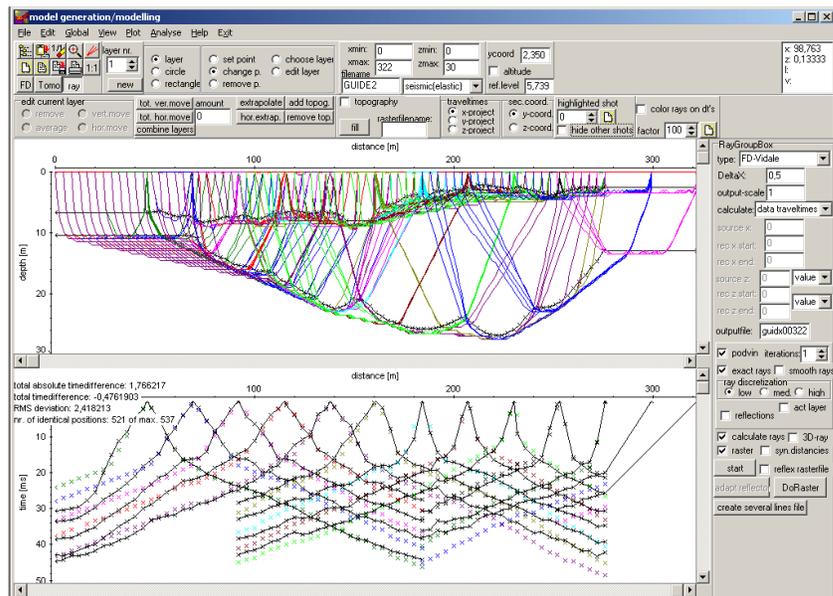
7. **Start** the raytracing.

8. At the end of the raytracing the calculated traveltimes are shown in the lower picture in addition to the observed data. Depending on which view option is checked, the calculated rays are shown in addition in the upper picture.

Now you may check for the mean traveltime difference using the option Analyse/**calculate traveltime differences using actual coord.projection**. As the positions of the data and the synthetic traveltimes may vary because of the fixed raster increment of the raytracing you may enter a position bin which is used for the determination of quasi-identical positions. By default the current raster increment of the raytracing is used.
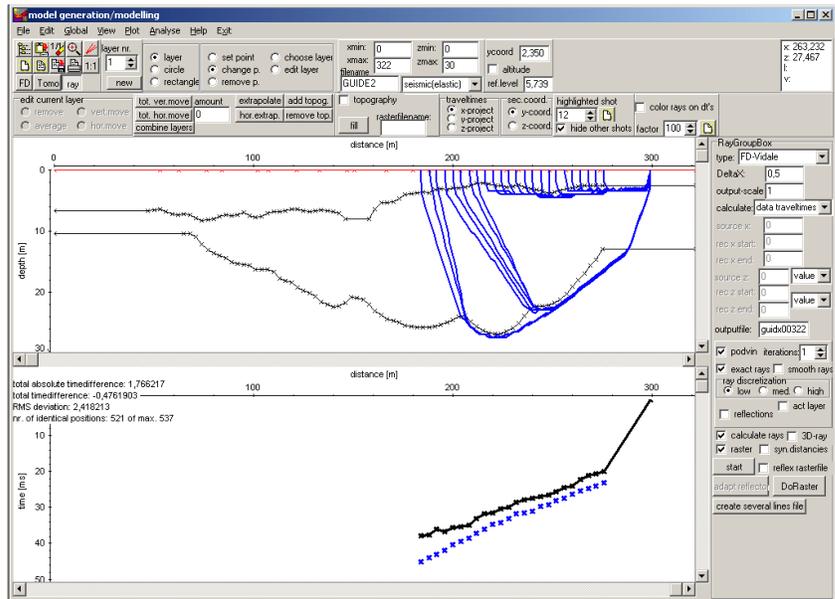
9. If the calculated and the observed traveltimes do not match, you may make some changes within the model and restart the raytracing in order to get a better match. For this purpose you can concentrate on one single shot using the option **highlighted shot.**, e.g. shot 12.
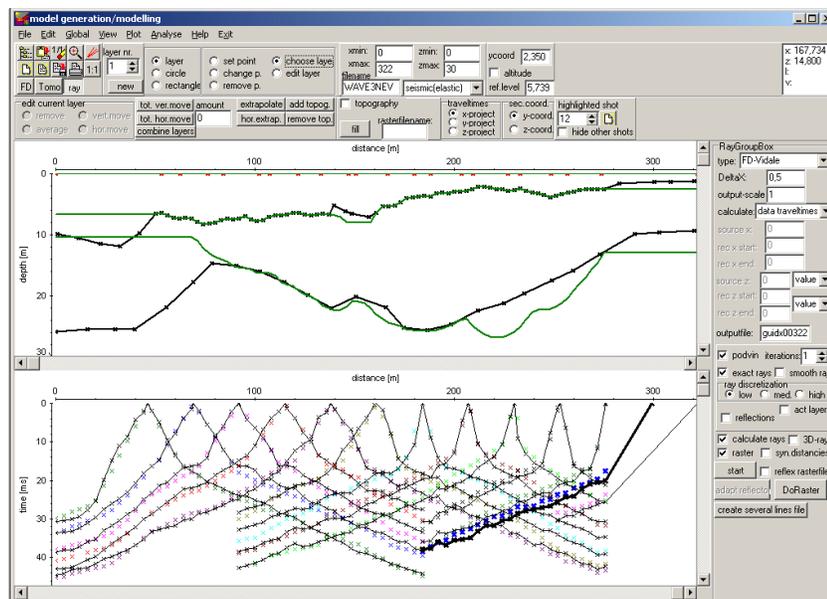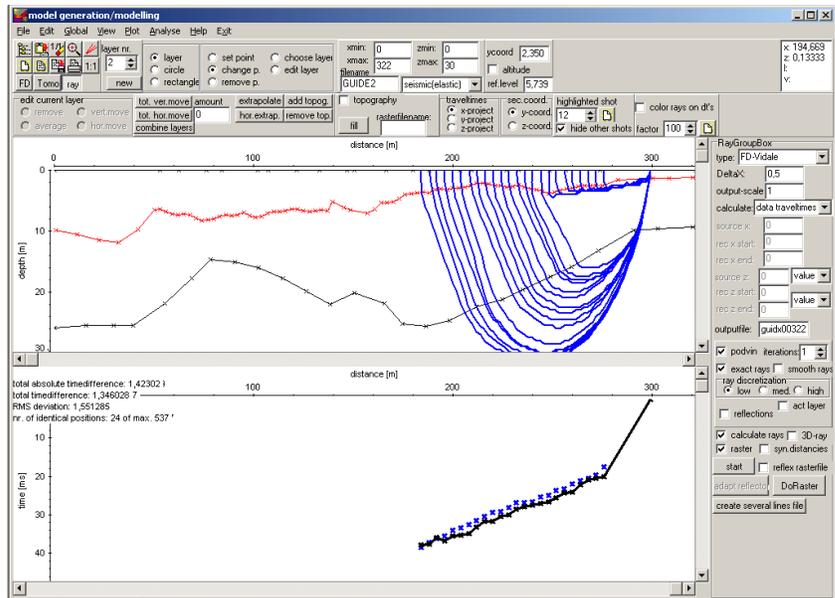
10. The last picture shows the final model (black) resulting from the interactive refining of the first model (stemming from the inversion of the observed traveltimes, green) by forward ray tracing and comparing the calculated with the observed traveltimes. As you can see the main differences are at the outer borders of the model because these cannot be directly inverted because of the missing receiver or shots points there.

IV. Refraction-tomography: A second kind of interpreting the data

In the case of the **2D refraction vertical tomography** all sources and receivers are located within one line at the surface. In order to allow for a high data coverage within the medium vertical velocity gradients should be present and a curved raytracing for the calculation of the traveltimes must be used. The curved rays are calculated using a finite difference approximation of the eikonal equation . A start model must be defined. No assignment to layers is necessary.

The start model should contain a quite strong vertical velocity gradient and the max. velocity variations for the tomographic inversion should be large enough (e.g. 200 % of the original values) in order to enable strong vertical gradients at those positions where an interface is assumed. A smoothing in horizontal direction is often useful because of the normally quite large receiver increments.

1. First a starting model must be generated or an already existing model must be loaded using the option file/load model. Enter the min./max. borders (xmin, xmax, zmin, zmax) in such a way that all desired shots and receivers positions fall into and that zmax exceeds the expected max. reached depth. The wavetype must be set to seismic(elastic) or acoustic. Normally the starting model is a simple homogeneous model with a quite strong vertical velocity gradient (dv/dz = 100 m/s per m, e.g.), whereby the velocity at the surface boundary should be within the expected range. In the following we will show two models which are generated based on the same starting model but different starting velocities: v = 800 m/s and v = 300 m/s.
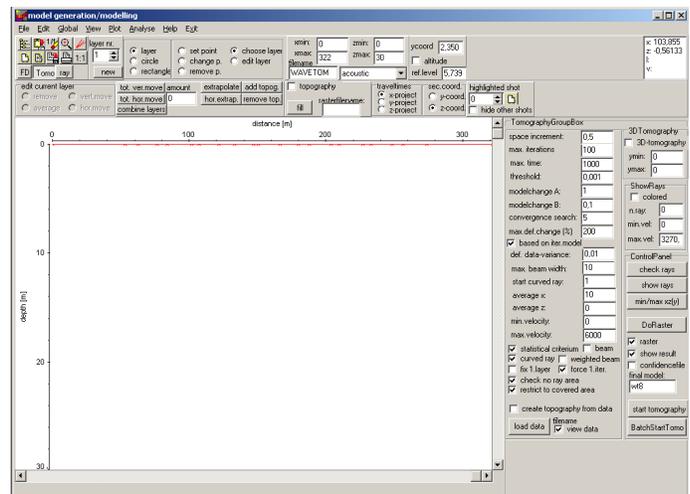
2. Activate the option **Tomo**

3. The **TomographyGroupBox** opens in addition (see figure above). Within this group box you have to enter the necessary tomography parameters.

- Load the data (ASCII-data with the extension tom) using the option **load data**. If the data are only available as pck file use the option export to Ascii within the traveltime analysis module in order to generate a tom file.
The program automatically controls, whether the data is in 2D- or in 3D-format.

- Enter the wanted **space increment** (equal in x- and z-direction, we used 1 m). Normally this increment should be small enough in order to allow small scale variations with depth. It should be significantly smaller than the receiver increment.

- The following options must be set for the refraction-tomography:
  - activate the option **curved ray**.
  - set the parameter **start curved ray** to 1.

- Enter a quite large value for **max.def.change (%)**. We used 200 %.

- Often it is useful to force the first iteration (option **force 1.iter.** activated) to generate a new model e-ven if the resulting residuals are larger than for the starting model.

- Enter a smoothing value in x-direction (parameter **average x**, we used 10, about one half of the shot-point distance).

- To minimize artefacts at the borders of the model and to be able to compare the resulting model with the result of the inversion, it is often useful to activate the option **restrict to covered areas**.

- Activate the option **show result** in order to display the tomography result.

- For a first tomographic result you may use the other default parameters. There are no general rules for these parameters. You have to adapt the parameters to your data in order to get the best result.

- Enter a name for the final model. Note: do not use the same name like for the starting model.

- **Start** the tomography. The tomographic result is stored using the "normal" REFLEXW format. You may display the result within the 2D-dataanalysis.
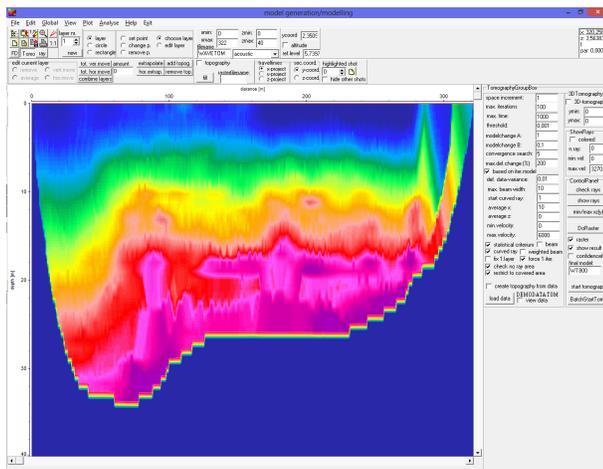
v = 800 m/s
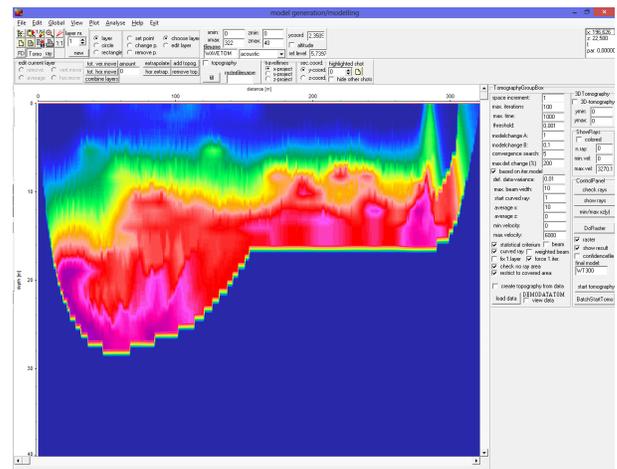
v = 300 m/s

4. The tomographic result can be controlled by a forward raytracing in the same manner as the inverted model. For that purpose activate the option **ray**. The raytracing menu opens in addition.
Load the traveltime data using the option File/**load data traveltimes**. Then the screen is split vertically showing in the upper window the model together with the tomographic result and in the lower the data. Now the ray tracing parameters have to be chosen:
- enter the wanted raytracing **type** FD-Vidale.
- enter the gridding increment **DeltaX** - this increment must be equal to the increment used for the to-mography (1 m in our case).
- enter the **output-scale**, e.g. 1
- enter the **calculate** type - in this case data traveltimes because we want to simulate all loaded observed traveltimes
- enter the **outputfile** name
- deactivate the option **raster**
- start the raytracing using the option **start**. As the option raster is deactivated you are asked for the ras-ter file. Choose as datatype reflex-files and choose the tomography raster file from the path rohdata.
- the calculated traveltimes are shown in the lower picture in addition. Now you may check for the

mean traveltime difference using the option Analyse/**calculate traveltime differences**.

Note: RMS deviations < 2 ms are acceptable for traveltimes < 100 ms and depth < 30 m, resp.. So, the tomographic result with both starting velocities are trustworthy.
Both resulting models show on their right edge (x > 275 m), that a reliable result can only be achieved, if information is existent not only from far distance shots. Otherwise the lack of information leads to artificial and hence unrealistic results.
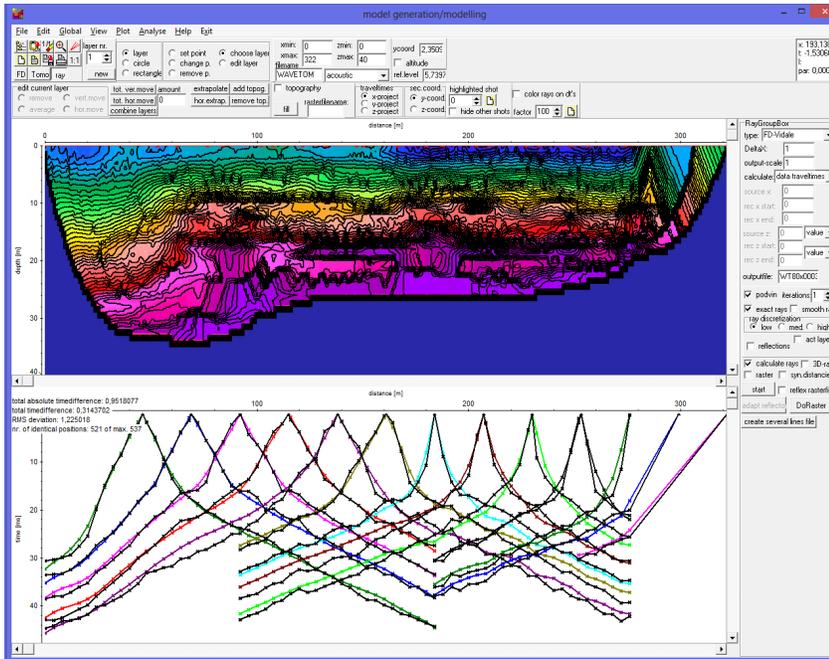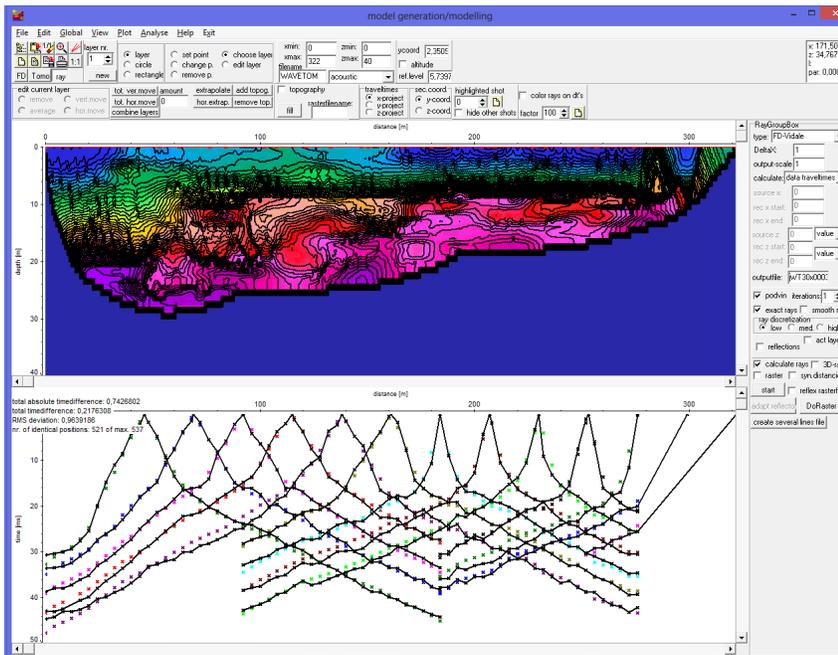
v = 800 m/s

The lower starting velocity yields to a "sharper" first layer boundary, whereby a general rule is confirmed: The tomographic algorithm works best, if the starting velocity is not to high and the velocity gradient is sufficiently strong, so that great velocity steps are possible. In addition, the gridding increment should not be to small to avoid artefacts.

In contrast to the model resulting from the wavefront-inversion, a shallow low velocity zone appears in the region x = 150 m to x = 280 m in both models resulting from the refraction-tomography, which is best visible, if isolines are plotted in addition. Especially the model with the starting velocity v = 300 m/s strongly suggests, that the data in the region x = 150 m to x = 280 m should be interpreted as an anthropogenic filling, i.e. an area with a strong vertical velocity gradient (v ≈ 500 m/ns up to v ≈ 1000 m/ns), instead of a rising boundary with little velocities (v ≈ 400 m/ns up to v ≈ 500 m/s), as the wavefront-inversion model suggests.

v = 300 m/s

Presentation of the tomographic result:

The tomographic result will be stored as a Reflexw file and therefore all processing possibilities within the 2D-dataanalysis are available. The presentation of the tomographic result may be improved using the following two processing steps:

- expand (e.g. using 4 in both directions, set the option keep 0 values to 1)
- median xy-filter (e.g. using 8 in both directions)

These processing steps smooth the stepwise character at the model borders due to the rough increment during the tomogprahic inversion - see picture below (original upper panel, processed lower panel)

The plotoptions have been set in the following way:

activated option AutoInterpolation and no interp.f.min.color

activated option background for 0 value

Activated options for isolines as shown within the box isolines (see image on the right)
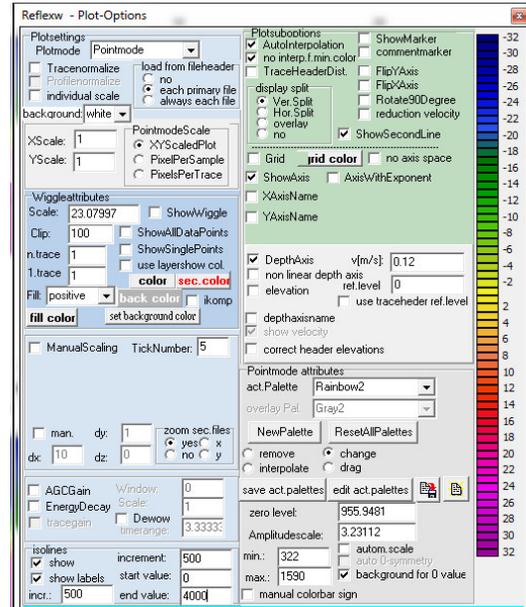
1. F:\data\luis\ROHDATA\tomo_05_100.DAT / traces: 122 / samples: 41
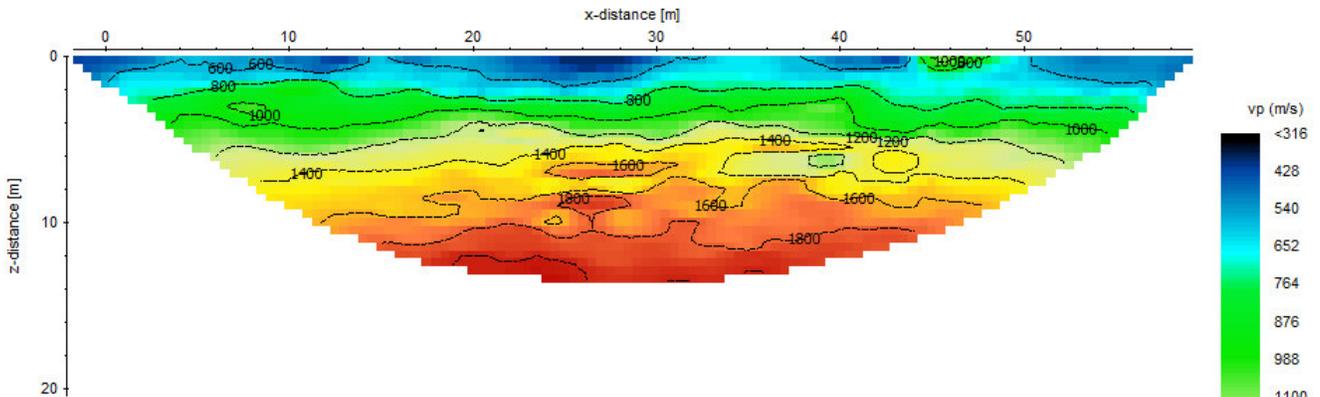
2. F:\data\luis\PROCDATA\TOMO_05_100.01T / traces: 485 / samples: 161
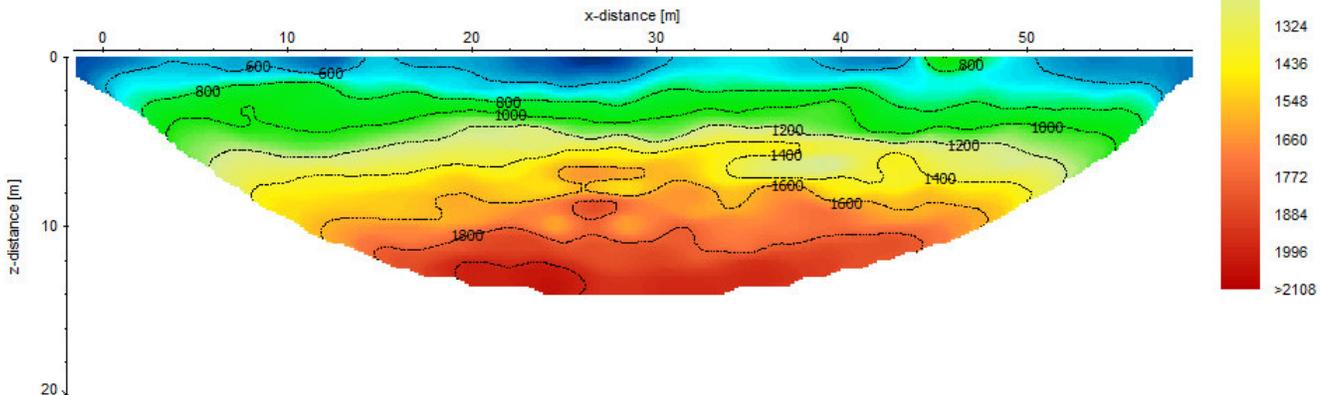
V. Joint interpretation of the results of the Wavefront-Inversion and the Refraction-tomography

As shown above, both methods provide different information about the underground and hold different sources of error, too: While the wavefront-inversion requires an assignment of data points to distinct layers, which is often not easy to decide, the refraction-tomography uses all information given, without paying attention, if the data coverage is good enough or the starting velocity leads to a reasonable result, e.g.. Therefore, we often recommend to take into account the results of both methods to come to a final interpretation of the data, which contains both, the information of the wavefront-inversion and of the refraction-tomography.

V. a Manual change of the wavefront inversion model under consideration of the tomographic results

In a first step it is often helpful to display both models together:

1. Enter the module **model generation/ modelling**.

2. **Load** the resulting model of the wavefront-inversion using the option file/load model.

3. **Load** the resulting model of the refraction-tomography (reflex-files format in folder rohdata) using the option view/show additional rasterfile. (We used the model with the starting velocity v = 300 m/s.)
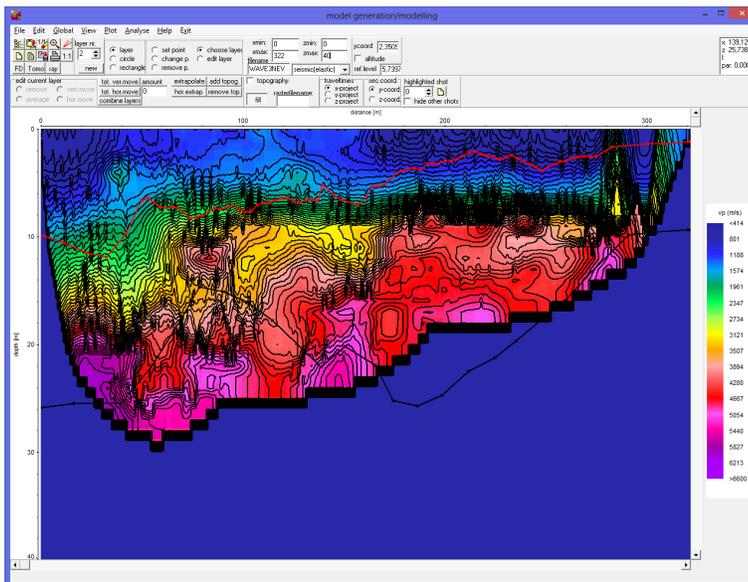
First of all it is visible in our example, that the second layer boundary is relatively undetermined due to the high velocities (v > 4500 m/s), but the first layer is represented quite well by both models in the region x = 0 m to x = 150 m.
For the region x = 150 m to x = 280 m our example illustrates impressively, which advantage the joint interpretation offers: The assumption of a distinct layer boundary stemming from the wavefront-inversion without taking into account the tomographic model would lead to an interpretation, which could result in serious consequences for buildings potentially constructed on top of this area! Not until with the help of the tomography model with it's strong velocity gradients this near-surface region can be interpreted as an anthropogenic filling, which leads – with all consequences for potential construction works – in greater depths than the sharp layer boundary resulting from the wavefront-inversion.

Taking this into account the primary wavefront-inversion model, which was generated without knowing the results of the refraction-tomography, can be analyzed once again as described in chapter III to obtain a refined model, which also maps the anthropogenic filling. The main changes concerned a strong vertical gradient has been included within the first layer between 150 and 280 m and the depths of the second layer which had been increased within this distance range.

The leftt of the following two figures shows the refined model (black) in comparison to the primary

wavefront-inversion model (green): The refined model fits the data just as good as the primary model. As can be seen in the right figure, the first layer boundary of the refined wavefront-inversion model fits the zone of the narrow isolines of the refraction-tomography model, i.e. the zone with very strong gradients, very well now.

So, the refined wavefront-inversion model does map the anthropogenic filling as well.
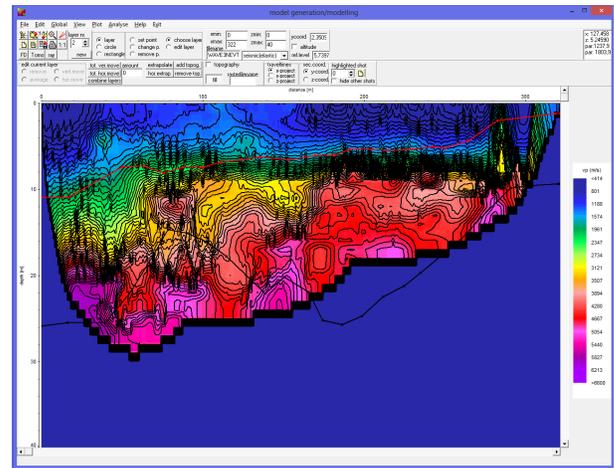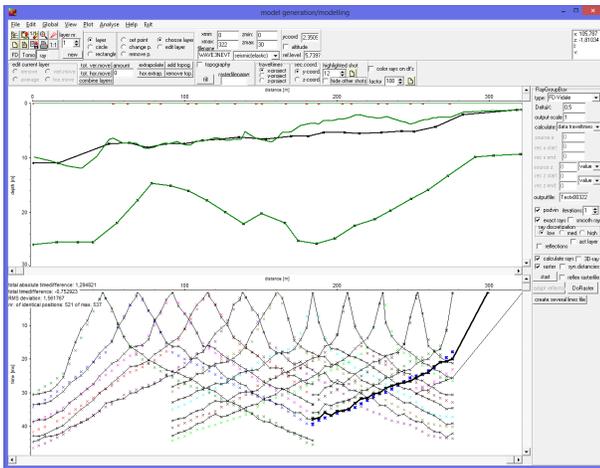
V.b Using the Wavefront-Inversion model as a starting model for the Refraction-tomography

To include the information gained from the wavefront-inversion into the refraction-tomography one can also use the primary wavefront-inversion model (ref. to chapter III) as a starting model instead of a homogeneous starting model with a quite strong vertical velocity gradient (left figure below).
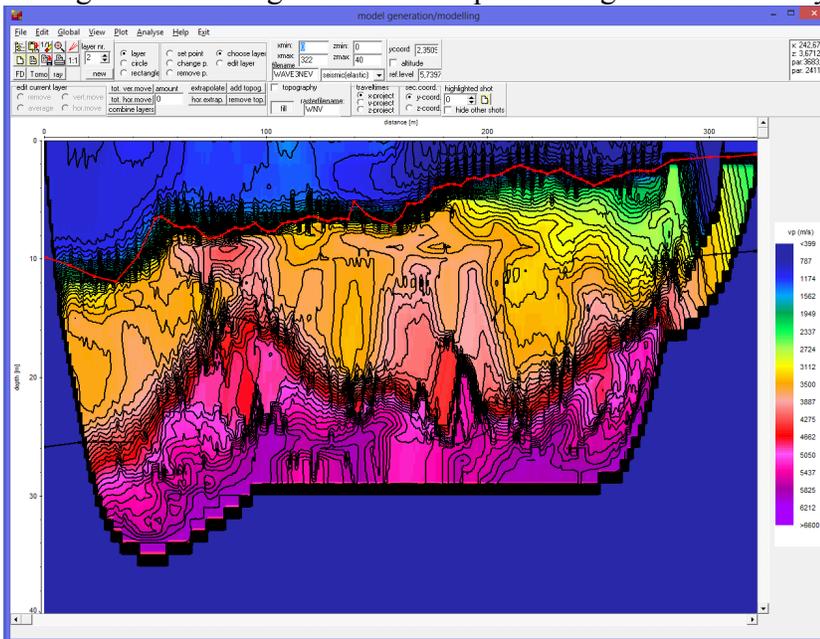
As expected, the tomographic algorithm – as a consequence of the much more restrictive starting model – is forced to provide a resulting model which reflects the two layer model resulting from the wavefront-inversion.

This is clearly visible eyeing the second layer boundary, which is now determined very distinct. But, since the restriction of the starting model is so strong, there is nothing else for the tomographic algorithm but to layaway along the predetermined layer boundary! Nevertheless, the resulting model – as the tomographic model with a homogenous starting model shown above – contains in the region x = 150 m to x = 280 m also a near-surface zone with relatively low velocities, which leads in greater depths than the sharp layer boundary resulting from the wavefront-inversion. With regard to potential construction works the two models resulting from the tomography both add up to the same result for this area: The basements of potentially constructed buildings have to be grounded in a greater

depth than indicated by the resulting model of the wavefront-inversion.

VI. Topography

The wavefront-inversion and the refraction-tomography do not automatically take into account the to-pography of a profile regarding it's localization in a given xz-coordinate system .
Normally the seismic refraction data are acquired along a line with equidistant distances on the topographic surface. These values are entered into the file and traceheader coordinates of the original seismic data. You should always use x as the profile direction and one value for the offset for all receivers and shots which shall be interpreted together.
For many cases it is not necessary to take into account any topographic xz-values (shot and receiver positions and elevations, resp.). For example, if the data is collected on a slope inclining with $\alpha = 10°$, the velocity of the bedrock is $v_b = 500$ m/s and the topography is not taken into account – which means that the geophone-distances d_g are taken as correct x-coordinates and the elevation is neglected – the apparent velocity is $v_a = 508$ m/s ($d_g = x / \cos \alpha$ and hence $v_a = v_b / \cos \alpha$, $1/\cos 10° = 1,015$), which means a tolerable increase of the velocity $I < 2 \%$.
But, if the inclination is stronger, it has to be taken into account to avoid significant errors:
$\alpha = 25°$, $1/\cos 25° = 1,103$, $v_b = 500$ m/s: $v_a = 551$ m/s, $I > 10 \%$.

### VI.a Redefinition of the source and receiver geometries

If a strong topography is present it is important to work with geometries based on a true xz-coordinate system. If the data geometry of each trace and thereby each traveltime is already given as **xz-coordinates**, e.g. using GPS no redefinition is necessary and you can skip this chapter.

If the data have been acquired along the **topographic interface** and the z-values along the acquisition line have been acquired independently, e.g. using GPS first the geometries of the shots and the receivers must be redefined, whereby the following preconditions must be fulfilled:
- The x-values of the current traveltimes (before redefinition) **do not** represent the correct x-coordinates but are determined directly on the topographic interface.
  For example, the data is collected with a fixed receiver offset = 2 m along a line with a variable inclination. So, the measured x-values would be x = 0 m, x = 2m, x = 4 m, ..., which do not coincide with the **true** x-coordinates within the xz-coordinate system .
- The topographic xz-values can be read from an ASCII-file whereby each line of the ASCII-file contains one pair of xz-values. The x-coordinate within the ASCII-file represents the **true** x-coordinate within the xz-coordinate system (stemming from GPS-measurements, e.g.). The z-values are either depths or altitudes.
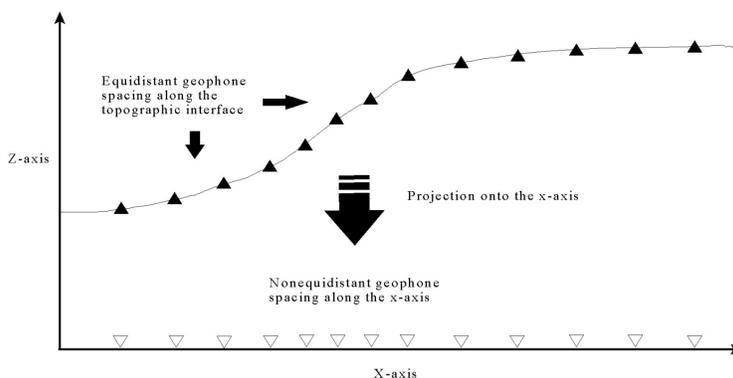
With the help of this ASCII-file the program automatically determines the positions of all shots and receivers on the given topography and calculates the x- and z-projections of these positions: The geometries of the shots and receivers are redefined and can be stored using a new file name. This redefinition is necessary for big slopes. For a slope of 10° the redinition is in the range of 1 % and therefore negligible but for a slope of 40° as within the lower example the x-projection is only 75 % of the topographic distance and therefore not negligible any more.

The redefining of the geometry of a profile – and hence the consideration of the topography regarding it's localization in a given xz-coordinate system – is done in the module **traveltime analysis.**

1. Enter the module **traveltime analysis**.

2. **Load** the wanted traveltimes which shall be interpreted together - option file/load traveltimes - multiple choice using the shift- or ctrl-key.

In order to display the data in 'refraction mode' (i.e. the time axis faces upwards) please activate the plot option **FlipYAxis**. If the option **colored** is activated, every shot is displayed in a certain color. This may help to get a first overview of the picked traveltimes.

3. **Star**t to redefine the geometry of the positions of the shots and the receivers using the option edit/apply x-z topography. If **convert altitude to depth** has been activated the altitude values within the ASCII-file will be converted to depth values from the difference of the entered reference level and the altitude values (reference level - altitude values). The reference level should be set at least to the maximum existing altitude value to achieve positive depth values.

4. Pressing the start button opens a window allowing to choose the ASCII-file within the folder ASCII containing the topographic xz-values.

5. Choosing the file automatically starts the redefinition of the geometry: The positions of the shots and the receivers move closer to each other, why the gradients of the traveltime branches increase and hence the velocities decrease (see straight lines in the figures).

6. To store the data with redefined geometry enter a **new** name and store the data. Otherwise the original data will be overwritten.

7. The resulting traveltimes now contain both the x- and z-coordinates. You may check the geometries using the option file/export to ASCII.
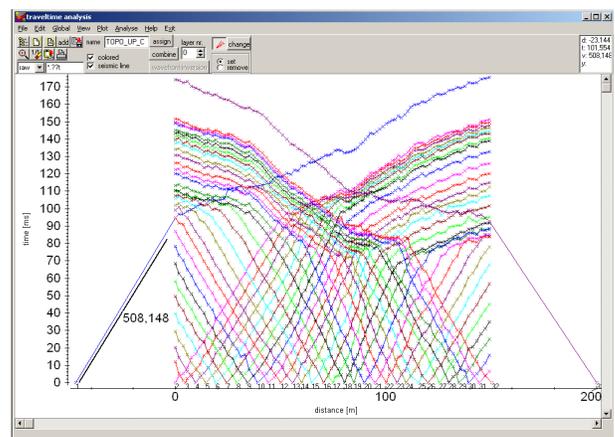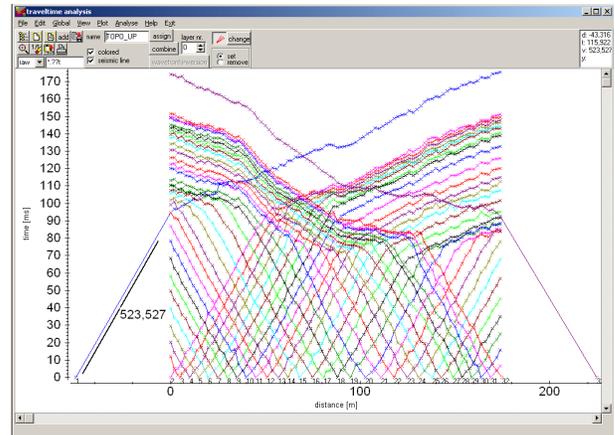
VI.b Considering the topography

There are two possibilities–both of them having advantages–to take the topography into consideration: Firstly, one can do the modelling by disregarding the topography until the final model is found and simply adding the topography thereafter. In many cases (see above), acting like this provides results with negligible errors but makes the data analysis much more easier: Please refer to chapter VI.b1 "Easy" Topography.
Secondly, the topography is taken into account creating the start model, which is always the correct way: Please refer to chapter VI.b2 "Correct" Topography.

VI.b1 "Easy" Topography

The "easy" topography has some advantages. The most important is the significantly lower computing time, because of the smaller depth (i.e. zmax) of a model, if the topography is not considered.
Furthermore, the manual adaptation of the model is simpler, because the complete (smaller) model is better visible on the monitor screen and the sloping or the rising, resp., of a layer boundary can be recognized a lot easier, if the topography is not taken into account.
The 'easy' model is achieved by firstly doing the inversion according to chapter II, i.e., the topography is taken into account not at all. To be considered: if the geometry has been redefined according to chap. VI.a you must be careful with the derived velocities of the upper layer. These velocities are too small for big layer slopes. You must manually correct the velocities using the cosine of the slope (see also estimate at the beginning of the chapter).
After having done the inversion, the topography is simply added in three steps using the modelling module:

1. Change **zmax** in such a way that the complete model including topography fall into this range.

2. **Select the 1. layer** and **use the option import (x,z)** within the Input of model parameters window to import the topographic xz-values from an ASCII-file. The ASCII-file may either only contain x- (=distance) and z-coordinates or x-, y- and z-coordinates. In the second case the distance along the line will be automatically calculated from the x- and y-coordinates. The first given (x,y) coordinate pair must correspond to the start (min.) position of the model.
The geometry of the first layer will be changed according to these topographic x(distance)z-values. The option topography will be automatically activated if deactive. If the values within the ASCII-file exceed the max. x and z model values the borders of the model will be expanded accordingly.

3. If the first layer boundary is not fully determined over the whole model range use the option **hor. extrap.**

4. As the first layer has a topography now, this topography can be added to ALL other layers by clicking ONE time the button **add topog.**.

As written at the beginning of this chapter, this "easy" method of adding the topography described here is o.k., whilst there are no regions with steep inclinations along the topographic surface. (On this refer to the last figure of the chapter VI.b "Correct" Topography, in which both methods are compared.)
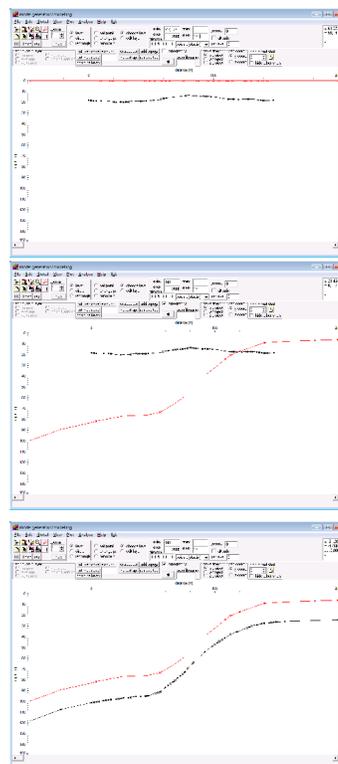
VI.b2 "Correct" Topography

1. Enter the module **traveltime analysis**. The wavefront-inversion is done according to chapter II, but the topography is taken into account now (see step 6 below):

2. Increase the **layer-no** to 1.

3. Activate the option **assign** in order to assign the wanted traveltimes to layer 1. (It is recommended to deactivate the option **colored** now.)

4. Assign the traveltimes to layer 1 by using the different possibilities. Use the options **Forward**, **Reverse** and **keep last shot** for a better definition at those dataparts, where the pick position is not well defined (e.g. overlapping files) - for further information refer to the online help. The traveltimes assigned to layer 1 will be highlighted (default color green), save the traveltimes.

5. Activate the option **combine** in order to do the inversion for the first layer. (To be considered: The inversion for the first layer only consists of the determination of the velocity.)

6. **Select Topography and/or altitude** in the CombinePanelLayer1 to take the topography into account correctly. (If the option altitude is activated in addition the z-values stored within the traveltimes define altitudes and no depths. The depth values are then calculated in the form reference level - altitude values. The reference level should be set at least to the maximum existing altitude value to achieve positive depth values. Please consider: If the option convert altitude to depth had been activated within step 3 the altitude/depth conversion has already been done and altitude should not be activated within this step.)

7. Activate the option **wavefront-inversion** and a new window (the modelling window) opens and a model has been automatically created consisting of one layer with the velocities taken from the linear regression analysis of the assigned traveltimes and the topography taken into account.


8. Do some changes of the model if you want (e.g. remove some unwanted velocity points, adapt the model size) - to be considered: the option topography should be activated. Increase zmax in such a way, that all layers to be inverted fall into this depth range.

9. Enter a modelfile name and save this model on disk.

10. Close the modelling window.

11. Increase the **layer-no**. to 2 and process the data according to chapter II until all the data is inverted.


The inverted model of the data shown here consists of two different layers, in which the topography is taken into account correctly.
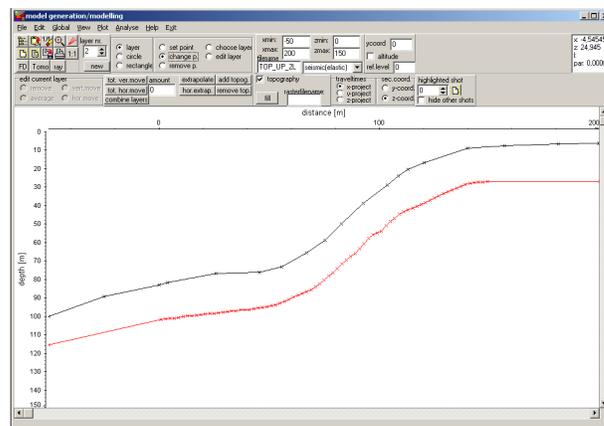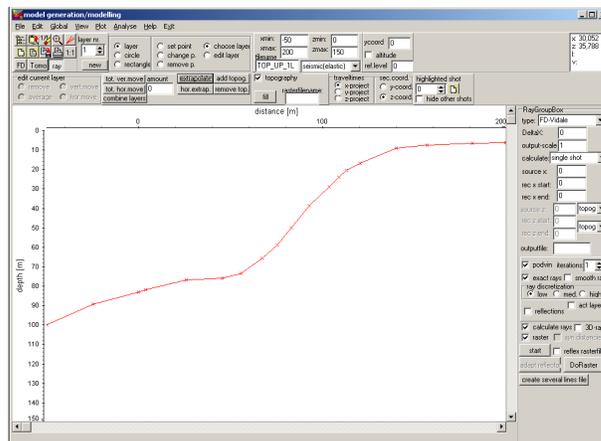
Which not negligible errors occur in our example, if the topography is not taken in to account during the inversion but is only added afterwards (refer to chapter VI.a "Easy" Topography), is shown in the last figure.
Looking at the both results displayed together, it is obvious, that in this region, where the inclination is steepest, the second layer boundary of the "easy" model (green line) differs up to 10 m (i.e. > 15 %, which is not tolerable!) from the correct model (red line).

As expected, the models do not differ significantly in that regions, where the inclination is not as steep.
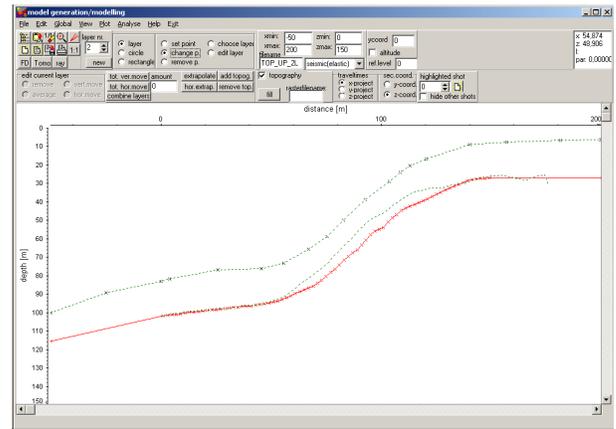
So, if there are no regions with steep inclinations along the topographic surface, the topography can be taken in to account in an easy and fast manner as described in chapter VI.b1 "Easy".
But the – admittedly more time-consuming – method, which leads always to a correct model, is to use the topographic xz-values to redefine the geometry before doing any inversion of the data.