Introduction to the modelling/tomography tools within REFLEXW

In the following the different modelling/tomography tools within the modelling menu of REFLEXW are described. The main tools are:
2D and 3D Finite difference modelling for seismic and electromagnetic wave propagation (chap. II)
2D ray tracing (chap. III)
2D- and 3D- transmission and refraction tomography (chap. IV)
Chap. I includes the generation of a model. This chapter is a more general one because all tools need a model to be generated first.
Please use in addition to this user's guide the handbook instructions and the online help.

I. model generation

First the generation of a complete new model is described.

1. Enter the module **modelling**

2. Choose the **wavetype** (e.g. electromagnetic for a GPR-modelling or acoustic for a raytracing).

3. Enter the **min./max. borders** of the model (parameters xmin, xmax, zmin, zmax). To be considered: z is going from top to bottom with positive numbers (e.g. zmin=0 and zmax=20).

4. activate the option **new**. The layer nr. is changed to 1.

5. The first boundary at z=0 has been automatically created by setting two layer points at the left and right upper corner point of the model. Now you must set the parameters for this boundary within the **input of model parameters window** which has been automatically opened. If this window is not on front press the right mouse button in order to bring it to the front.

6. You may set or change the parameters either by editing the corresponding fields of the table or by using the general fields situated in the right upper corner.
Using the second possibility first you have to enter the parameters within these fields. These parameters can be overtaken to selected points of the current layer. The selection is done by clicking on the fields in the first column of the table (the fields, which indicate the number of the points) and is indicated by a cross or by activating the option **take over all**. Clicking the **take over** button in the ControlBox leads to the updating of the parameters at the selected points or at each point (option take over all activated).

7. Use the button **update** for updating the model.
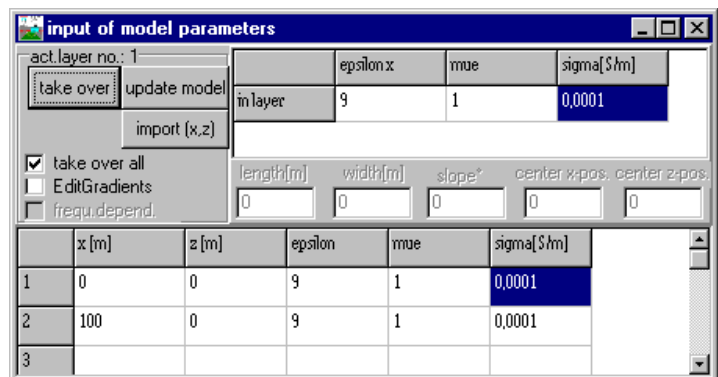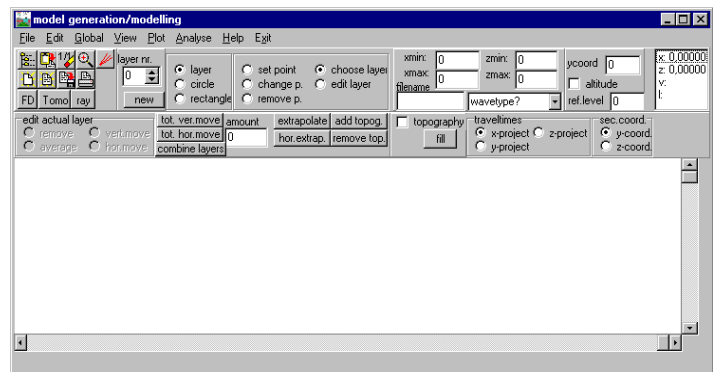
8. You may include some new layer points by simply clicking within the main modelling menu. The actual general model parameters of the fields within the right upper corner of the input of modelling parameter menu are automatically taken over for these new layer points. You may change these parameters like described under step 6.

9. Return to the main modelling menu by simply activating it or by closing the input of model parameters menu.

10. activate the option new for the next layer. The layer nr. is changed to 2.

11. For that layer you have to define all layer points and the corresponding model parameters - see step 6 to step 9.

12. Additional layers are defined as described for layer 2.

13. All layers must be closed - this means they must start or end at the model border or at any other layer boundary. It is not necessary to do this manually but the options extrapolate and hor.extrapol. may be used for that purpose. The interfaces don't have to be entered exactly at the edge, intersection with another interface, respectively, but in the vicinity because the option extrapolate makes the automatic interpolation between two interfaces as well as the extrapolation of the interface to the boundary possible. Therefore the program searches automatically the nearest border at the exposed end and extrapolates in that direction (in the case of an edge, extrapolation in x-, z-direction, respectively, in the case of an interface to the nearest layer point).

PLEASE NOTE: The layer points are sorted automatically from lower x-distance to higher x-distance, i.e. the interface function is unequivocal. Should for example a convolution be given it has to occur by giving several interfaces, like it is done by the predefined symbols circle and rectangle.

14. Enter a filename and save the model using the speedoption or the option file/save model.

15. For additional features like using predefined symbols, combining existing layers or adding a topography please refer to the handbook instructions under the individual chapters or to the online help.

II. Finite Difference (FD) modelling

The Finite Difference (FD) modelling tool allows the simulation of electromagnetic or seismic wave propagation, respectively by means of **FD-method** for different sources (plane wave, point source as well as "Exploding-Reflector"-source). As a result a single line or the complete wavefield are stored and can be displayed after.

In the following we are describing the GPR-simulation for a 2D zero-offset section (standard GPR-data acquisition).

1. First a new model must be generated (see chap. I) or an already existing model must be loaded using the option file/load model.

2. Activate the option **FD**

3. The **FD-GroupBox** opens in addition (see figure on the right). Within this group box you have to enter the necessary FD-Parameters.

4. First you must enter the main **frequency** for the simulation in Hz (seismics) or MHZ (GPR).

5. enter the wanted **source type** - for this example exploding reflector. The exploding reflector type allows you a simulation of a 2D zero offset section within one calculation.
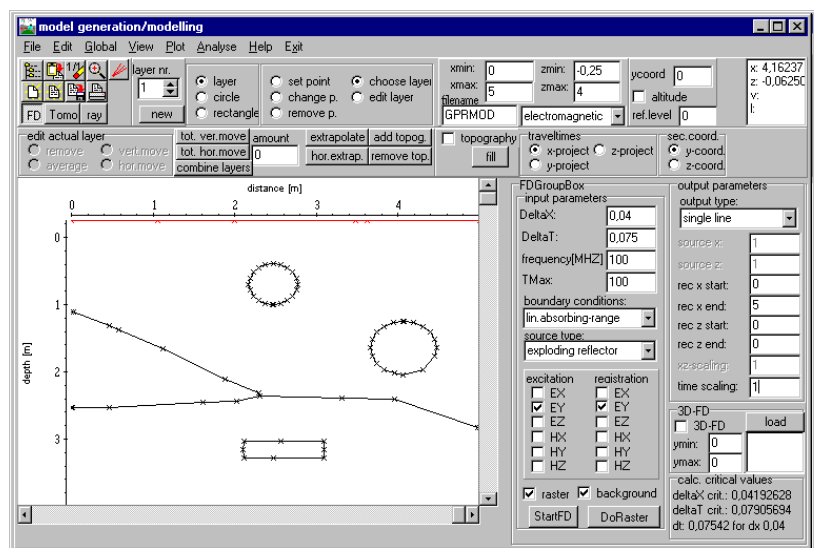
6. Enter the wanted **DeltaX** increment in space direction (equal in x- and z-direction). For the FD-computation the layer model has to be rastered with a given increment in x- and z-direction (option **DeltaX**). Just so, a time increment has to be given (option **DeltaT**). The size of the space- and time-increment corresponds to the minimal wave length as well as the velocity. The program determines automatically the critical value of the space-increment (1/8 of the min. wave length for FD-scheme 4-space, 1/12 of the min. wave length for FD-scheme 2-space, respectively) and shows this one in the calc. critical values box down right. This value should not be passed over. A too big chosen DeltaX increment results in numerical dispersion of the wavelet. Therefore if the result shows such a dispersion you have to decrease the DeltaX increment.

7. Enter the wanted timeincrement **DeltaT**. The max. time-increment depends on the max. velocity as well as on the given space-increment DeltaX (approximately: $\Delta t <= 1./(\sqrt{2}V)$ for the el.magnetic propagation and $\Delta t <= 1./(Vp+Vs)$ for the elastic propagation). The critical time-increment of the current set DeltaX increment is shown in the calc. critical values box at the bottom. A too big chosen DeltaT increment results in an instability, this means the amplitude increases exponentially with time. Therefore if the result shows such an amplitude increase you have to decrease the DeltaT increment.

8. Enter the total time **TMax** for the simulation.

9. Enter the **boundary conditions**, e.g. lin.absorbing range for the GPR-simulation.

10. Choose the wanted **excitation** and **registration** components. By default the EY-components are activated.

11. Choose the **output type** - for this example single line.

12. Enter the **output parameters** for the single line (rec x start, rec x end, rec z start, rec z end). For example: rec x start: 0; rec x end: 5; rec z start: 0; rec z end: 0 (registration line at the surface)..

13. Activating the option **StartFD** starts first the rastering of model (if the option **raster** is activated) and then starts the external program FDEMSEIS.EXE which will normally be executed in the background (option **background** activated), so that it is possible to go on with the work in REFLEXW. If the option <u>background</u> is deselected, the computation is faster but there is no possibility to work with REFLEXW until the FD-computation is finished.

14. After having finished the FD-computation you may display the simulation result within the 2D-dataanalysis.

III. ray tracing

The ray tracing modelling tool allows the traveltime simulation of electromagnetic or seismic wave propagation, respectively by means of a finite difference approximation of the eikonal equation. The calculation of the synthetic traveltimes is restricted to the first arrivals for an arbitrary 2-dimensional medium. No reflections and secondary arrivals can be calculated. This can be done using the FD-simulation (see chap. II).
The main application is the seismic refraction but it is also possible to simulate any transmission data. The raytracing may be used for
- the control of an inverted model
- an iterative adaptation of the calculated and real data by stepwise changing the underground model

In the following the application to seismic refraction data is described.

1. first a new model must be generated (see chap. I) or an already existing model must be loaded using the option file/load model.

2. activate the option **ray**

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

4. We want to simulate the observed traveltimes of different shots along the line. For that purpose we 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.


5. 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).
- enter the **output-scale**, e.g. 4
- enter the **calculate** type - in this case data traveltimes because we want to simulate all loaded observed traveltimes
- enter the **outputfile** name

6. Start the raytracing using the option **start**

7. 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**
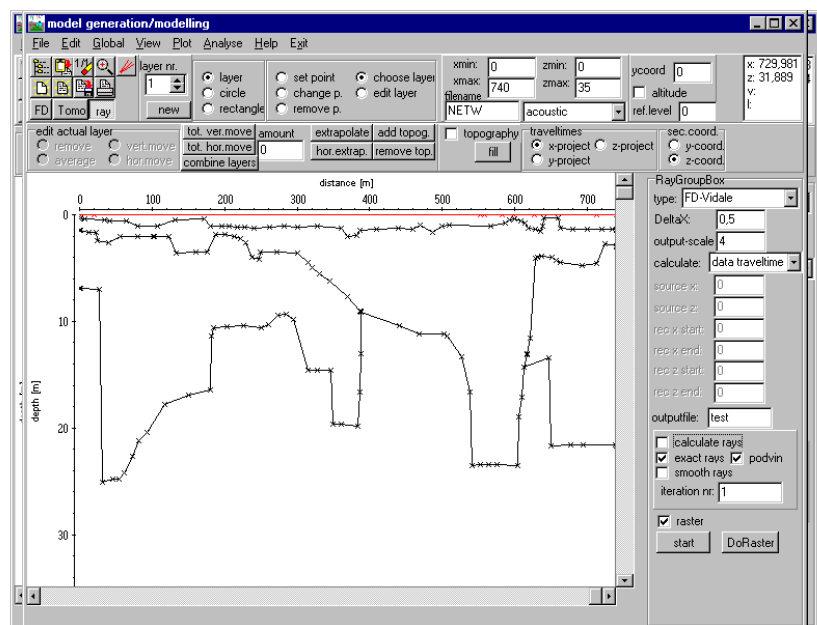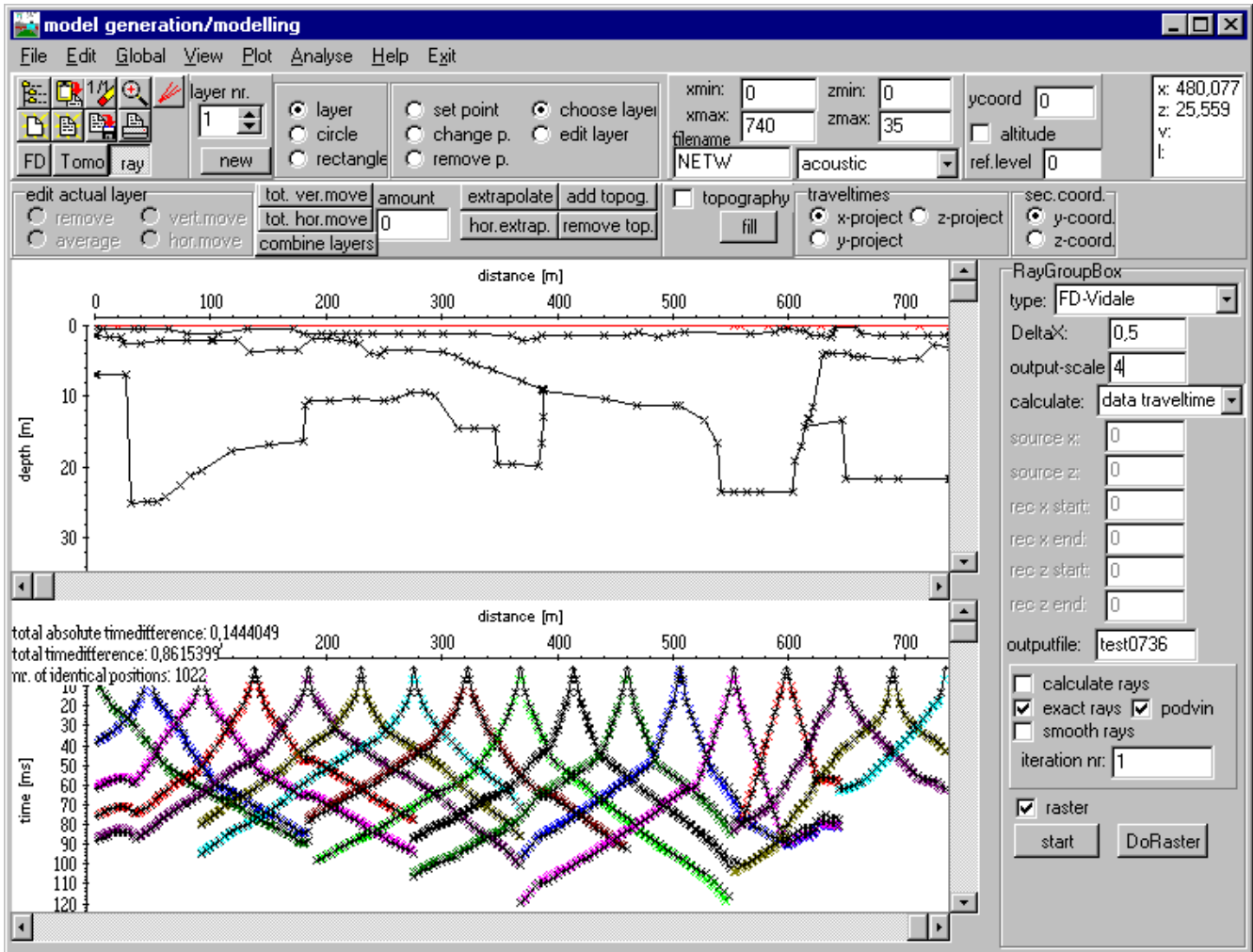
8. If the calculated and the observed raveltimes do not match you may make some changes wthin the model and restart the raytracing in order to get a better match.

IV. tomography

Tomographic methods have been well established for borehole-borehole or borehole-surface measurements whereby the object will be directly transmitted (so called **transmission tomography**).

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.

REFLEXW supports a 2D- and 3D-transmission traveltime tomography and a 2D refraction tomography. For the 2D-tomography both straight and curved raytracing is supported. For the 3D-tomography only straight raytracing is supported.

Chapter IV.1 includes the format and the picking of the **traveltime data**.
In chapter IV.2 the 2D tomographic interpretation of **borehole-borehole transmission tomography** is described.
In chapter IV.3 the 2D tomographic interpretation of a **refraction tomography** is described.

IV.1 picking the traveltime data and description of the format

Before performing the tomography the traveltime data to be inverted must be present.

REFLEXW uses a 2D or 3D ASCII-data format:

2D-data format:
travel time, code, transmitter_X, transmitter_Y(Z), receiver_X, receiver_Y(Z)
The code is given as an integer value.
example (2 travel times):
  800.00   1   0.00   2.00 100.00   2.00
  801.60   1   0.00   2.00 100.00   6.00

3D-data format:
traveltime,code,transm._X,transm._Y,transm._Z, receiv._X, receiv._Y, receiv._Z
example (2 travel times):
  800.00   1   0.00   2.00   1.00 100.00   2.00   5.00
  801.60   1   0.00   2.00   1.00 100.00   6.00   5.00

These data can be created externally or within REFLEXW when picking the original wavedata.

In the following the picking of the original wavedata within REFLEXW is described.

The most important part is the definition of the source- and receiver-coordinates. These coordinates are stored within the **traceheader** of each trace. There are two different possibilities for both the refraction and the transmission data.
Either each shot is imported (single shot analysis), filtered and picked separately (standard procedure for "normal" refraction interpretation - see also refraction guide, Import the data and pick the first

onsets (done within the module 2D-dataanalysis).
Or one datafile contains all shots (several shots analysis - standard procedure for the reflection seismic interpretation - see also reflection guide, Import the data and setting the geometry (both done within the module 2D-dataanalysis) and the data are filtered and picked in one step. If a very high data coverage is present the refraction data may also be handled like the reflection data with one datafile containing all shots. In that case the various possibilities of defining the geometry may be claimed.

To be considered: In any case the traceheader coordinates must be correctly defined before picking! If the 2D-data format shall be used the x- and y-traceheadercoordinates will be considered. The z-traceheader coordinates will be ignored. In the following the definition for this 2D-data format for equidistant receivers will be described.

IV.1.1 single 2D shot analysis

Dependeing on the shot/receiver geometry three different data types can be distinguished:

single shot: **surface/surface** geometry - shots and receivers are located on the surface, e.g. seismic refraction or reflection data
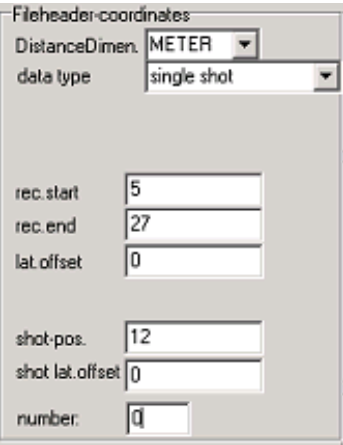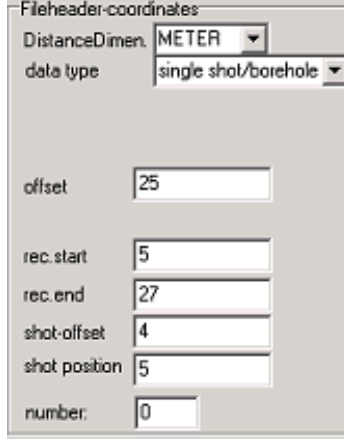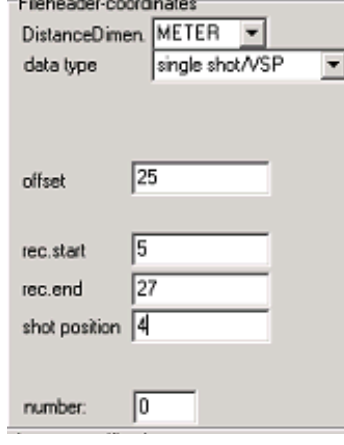**single shot/boreholes**: **borehole/borehole** geometry - shots and receivers are located within two parallel boreholes
single shot/VSP: **borehole/surfac**e geometry - the receivers are location within a borehole, the shots are located at the surface

The shotfiles can be imported and handled completely separately or using a conversion sequence.

IV.1.1.1 import each shotfile separately (conversion sequence set to no)

Enter the import menu and choose the correct data type and enter the overall 2D-fileheader geometry of your shot and the receivers- see the tabella below. Depending on your original data it might be useful to deactivate the import option read traceincr. Otherwise the end receiver coordinates might be changed.

surface/surface	Borehole/borehole	Borehole/surface
The equidistant receivers are located at the surface from 5 to27 m. The shot is placed at 22 m. The lateral offsets of the receivers and the shot are 0.	The equidistant receivers are located within borehole 1 (25 m along the surface) from 5 to 27 m depth. The shot is situated within borehole 2 (4 m along the surface) at 5 m depth.	The equidistant receivers are located within the borehole (at 25 m along the surface) from 5 to 27 m depth. The shot is situated along the surface at 4 m.

Fileheader-coordinates (surface/surface):
- DistanceDimen. METER
- data type single shot
- rec.start 5
- rec.end 27
- lat.offset 0
- shot-pos. 12
- shot lat.offset 0
- number: 0

Fileheader-coordinates (Borehole/borehole):
- DistanceDimen. METER
- data type single shot/borehole
- offset 25
- rec.start 5
- rec.end 27
- shot-offset 4
- shot position 5
- number: 0

Fileheader-coordinates (Borehole/surface):
- DistanceDimen. METER
- data type single shot/VSP
- offset 25
- rec.start 5
- rec.end 27
- shot position 4
- number: 0

If the traceheader coordinates shall be updated based on these overall fileheader coordinates choose **fileheader** for **update traceheaders** within the update traceheader/gps cordinates box. The receiver positions are assumed to be equidistant.

update traceheaders/gps coordinates
- update traceheaders: fileheader
- utm-conversion: no
- □ calculate distancies
- □ offset -> x
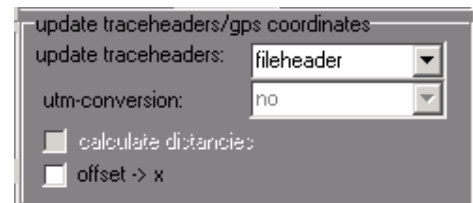
After having converted the original shotfile the edit traceheader coordinates tabella menu opens (example below for the borehole/borehole layout). If the traceheader coordinates are not correct there exist different possiblities to change them. They can be manually entered or load from an ASCII-file or you may use the option **update from fileheader** in order to update the traceheader coordinates from the entered fileheader coordinates if not already done(see above). By default the shotx and recx positions describe the location along the surface and the shoty and recy positions describe either the positions within the boreholes or the y-offsets. Close the menu with saving the changes. Now the geometry of this single shot is ready and the shot may be processed or the first arrivals may be picked. Do the same procedure for all subsequent shots.

edit trace header coordinates

trace-nr.	distance	shot-x	shot-y	shot-z	rec.-x	rec.-y	rec.-z	time delay	gain	time collect
1	5	4	5	0	25	5	0	0	1	0
2	7	4	5	0	25	7	0	0	1	0
3	9	4	5	0	25	9	0	0	1	0
4	11	4	5	0	25	11	0	0	1	0
5	13	4	5	0	25	13	0	0	1	0
6	15	4	5	0	25	15	0	0	1	0
7	17	4	5	0	25	17	0	0	1	0
8	19	4	5	0	25	19	0	0	1	0
9	21	4	5	0	25	21	0	0	1	0
10	23	4	5	0	25	23	0	0	1	0
11	25	4	5	0	25	25	0	0	1	0
12	27	4	5	0	25	27	0	0	1	0

| topography (x,z values)	load from AsciiFile	update from fileheader	source <-> rec.	check rec.coordinates		reload from file
☑ update shot z-pos.	save on AsciiFile	update fileheader	x <-> y	factor f.check: 10		save changes
☑ update receiver z-pos.						
☑ use x-traceheadercoord.		update distancies	project on x			
☐ apply x-z topography						
			interpolate			close

IV.1.1.2 import all shotfiles and set the geometry within one step (conversion sequence set to inline lines)

In all cases the conversion sequence **inline lines** can be used in order to import the different shot files within one step with an automatic actualization of the overall fileheader geometry and of the traceheader coordinates. For the latter case choose **fileheader** for **update traceheaders** within the update traceheader/gps cordinates box. After having activated the option convert to Reflexw you must select all original files which fall in the selected geometry layout using the shft or ctrl key. In the case of the converison sequence inline lines the edit traceheader coordinates menu does not open after having converted the files.
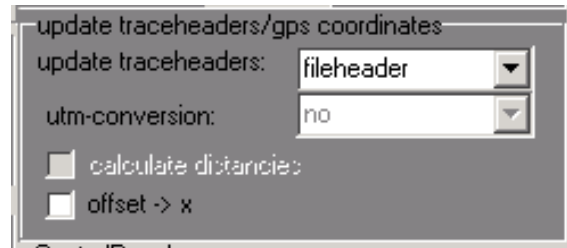
Surface/surface layout 1 with moving receivers and shots (e.g. seismic refraction or reflection data). For a fixed line with moving shots simply deactivate move receivers.
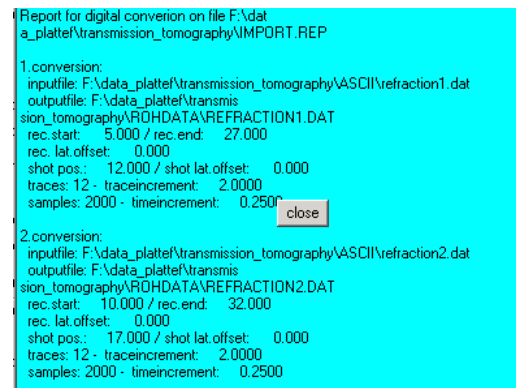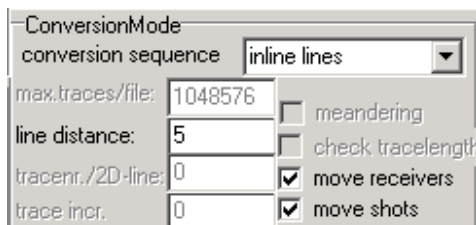
After having imported all files it is possible to redefine the geometry of the single shots within the edit several fileheaders menu (to be found under file):

- Select the wanted filepath and open files.
- Choose all wanted files and make any changes for the shot and receiver positions if necessary. Rec-start and rec-end as well as shot pos. describe the location along the surface. Rec-offs. and shot offs. describe the lateral offset.
- Choose "update trace headers" -> fileheader and click on save. You may check your geometry using the option file/edit traceheader.

Sandmeier geophysical software - REFLEXW guide
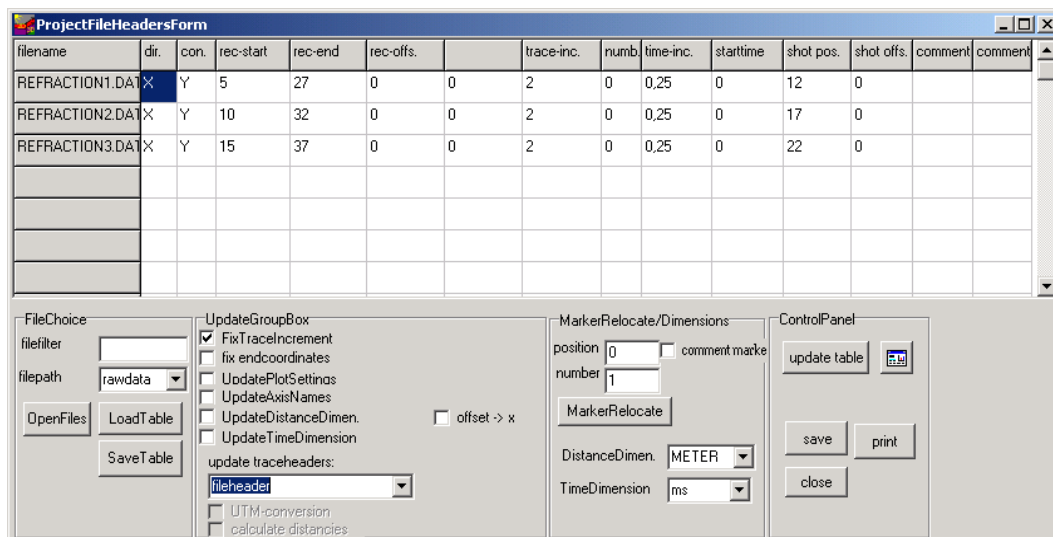
Borehole/borehole layout 2 with a start positionwithin the borehole for the shot at 15 m and moving shots upwards with an increment of 5 m:

ConversionMode
conversion sequence | inline lines

max.traces/file:	1048576
line distance:	-5
tracenr./2D-line:	0
trace incr.	0

☐ meandering
☐ check tracelength
☐ move receivers
☑ move shots

Report for digital converion on file F:\dat
a_plattef\transmission_tomography\IMPORT.REP

1.conversion:
  inputfile: F:\data_plattef\transmission_tomography\ASCII\5001.dat
  outputfile: F:\data_plattef\transmission_tomography\ROHDATA\5001____.DAT
  rec. offset: 25.000
  rec.start: 5.000 / rec.end: 27.000
  shot offset: 4.000 / shot position: 15.000
  traces: 12 - traceincrement: 2.0000
  samples: 2000 - timeincrement: 0.2500

[close]

2.conversion:
  inputfile: F:\data_plattef\transmission_tomography\ASCII\5002.dat
  outputfile: F:\data_plattef\transmission_tomography\ROHDATA\5002____.DAT
  rec. offset: 25.000
  rec.start: 5.000 / rec.end: 27.000
  shot offset: 4.000 / shot position: 10.000
  traces: 12 - traceincrement: 2.0000
  samples: 2000 - timeincrement: 0.2500

3.conversion:
  inputfile: F:\data_plattef\transmission_tomography\ASCII\5003.dat

After having imported all files it is possible to redefine the geometry of the single shots within the edit several fileheaders menu (to be found under file):

- Select the wanted filepath and open files.
- Choose all wanted files and make any changes for the shot and receiver positions if necessary. Rec-offs. and shot offs. describe the location along the surface. Rec-start and rec-end as well as shot pos. describe the location within the boreholes.
- Choose "update trace headers" -> fileheader and click on save. You may check your geometry using the option file/edit traceheader.

ProjectFileHeadersForm

filename	dir.	con.	rec-offs.		rec-start	rec-end	trace-inc.	numb.	time-inc.	starttime	shot offs.	shot pos.	comment	comment
5001____.DAT	Y	X	25	0	5	27	2	0	0,25	0	4	15		
5002____.DAT	Y	X	25	0	5	27	2	0	0,25	0	4	10		
5003____.DAT	Y	X	25	0	5	27	2	0	0,25	0	4	5		

FileChoice
filefilter [   ]
filepath [rawdata ▼]
[OpenFiles] [LoadTable]
[SaveTable]

UpdateGroupBox
☑ FixTraceIncrement
☐ fix endcoordinates
☐ UpdatePlotSettings
☐ UpdateAxisNames
☐ UpdateDistanceDimen.
☐ UpdateTimeDimension
update traceheaders:
[fileheader ▼]
☐ UTM-conversion
☐ calculate distancies

latitude
◉ x-coord
○ y-coord

MarkerRelocate/Dimensic
position [0] ☐ comm
number [1]
[MarkerRelocate]
DistanceDimen. [METEF]
TimeDimension [ms]

Sandmeier geophysical software - REFLEXW guide

Borehole/surface layout 3 with a start positionwithin the borehole for the shot at 15 m and moving shots upwards with an increment of 5 m:

Report for digital converion on file F:\dat
a_plattef\transmission_tomography\IMPORT.REP

1.conversion:
  inputfile: F:\data_plattef\transmission_tomography\ASCII\5011.dat
  outputfile: F:\data_plattef\transmission_tomography\ROHDATA\5011____.DAT
  rec. offset: 25.000
  rec.start: 5.000 / rec.end: 27.000
  shot offset: 4.000
  traces: 12 - traceincrement: 2.0000
  samples: 2000 - timeincrement: 0.2500

[close]

2.conversion:
  inputfile: F:\data_plattef\transmission_tomography\ASCII\5012.dat
  outputfile: F:\data_plattef\transmission_tomography\ROHDATA\5012____.DAT
  rec. offset: 25.000
  rec.start: 5.000 / rec.end: 27.000
  shot offset: 9.000
  traces: 12 - traceincrement: 2.0000
  samples: 2000 - timeincrement: 0.2500

3.conversion:
  inputfile: F:\data_plattef\transmission_tomography\ASCII\5013.dat

After having imported all files it is possible to redefine the geometry of the single shots within the edit several fileheaders menu (to be found under file):

- Select the wanted filepath and open files.
- Choose all wanted files and make any changes for the shot and receiver positions if necessary. Rec-offs. and shot pos. describe the location along the surface. Rec-start and rec-end describe the location within the boreholes. The shot is always situated at the surface at a depth of 0m.
- Choose "update trace headers" -> fileheader and click on save. You may check your geometry using the option file/edit traceheader.

It is only possible to redefine the geometry within one step for the same shot/receiver layout. Therefore if you have two different layouts as described within example 1 and 2 you must redefine the geometry for these two layouts separately.

Sandmeier geophysical software - REFLEXW guide

IV 1.2 several shots analysis

Putting all shots together has several advantages:
- setting the geometry is easier
- picking is faster

You can put together the shots during the import (conversion sequence combine lines/shots) or afterwards using a special processing option. The shots must have the same sample number if using the import option conversion sequence "combine lines/shots".

The step-by-step procedure:
1. **Import** your single shot data. Set datatype to several shots because in step 2 all shots will be combined within one single file. The order of the files must be correct (e.g. file001.sg2 and not file1.sg2) because otherwise the sorting of the files is not correct for step number 2.
If you are using the conversion sequence "combine lines/shots" it is possible to load all shots within one import step (multiple file choice using the ctr or shft key). The step number 2 can then be ignored. The coordinates must be defined afterwards if not already existing within the original data. Therefore choose **no** for **update traceheaders** within the update traceheader/gps cordinates box.

2. load the first shot and put together all shots using the option **combine files f. CMP** under processing/edit traces. Click on load and choose all the other shots except the actually loaded file (multiple file choice using the ctr or shft key). To be considered: the sorting of the files is done automatically with ascending alphabetic order of the filenames. Therefore a renaming of the files e.g. 1.dat,...,9.dat to 01.dat,...,09.dat may be necessary.

3. do any **filtering**, e.g. bandpassfiltering and time cut (optional).

4. enter the **geometry** using the option **CMP** (see also chap. 1.12.4.1). Click on geometry and activate fixed line for standard geometry. Enter the geometry within the standard geometry box.
The radio box standard line direction allows to define the direction of the standard geometry.
x-direction activated: the line (shots and receivers) is assumed to be orientated in x-direction (surface/surface layout). This is the case for a seismic refraction dataset. Shot offset and receiver offset define the constant offset in y-direction and should be set to 0 for seismic refraction data.
y-direction shots/rec. activated: the total line (receivers and shots) is assumed to be orientated in y-direction (borehole/borehole layout). Use this option for example to define the geometry of a two boreholes transmission measurement. Shot offset and receiver offset define the positions of the 2 boreholes along the x-axis (surface).
y-direction shots activated: the shots are assumed to be orientated in y-direction (surface/borehole
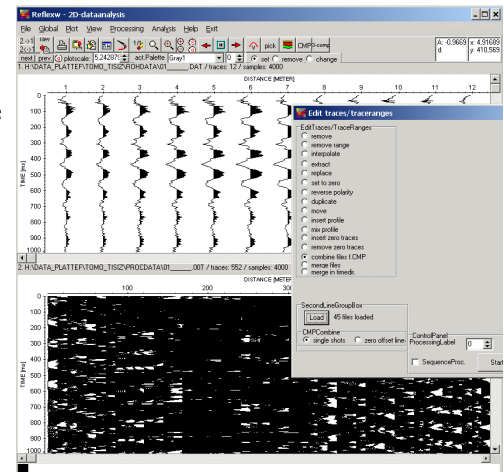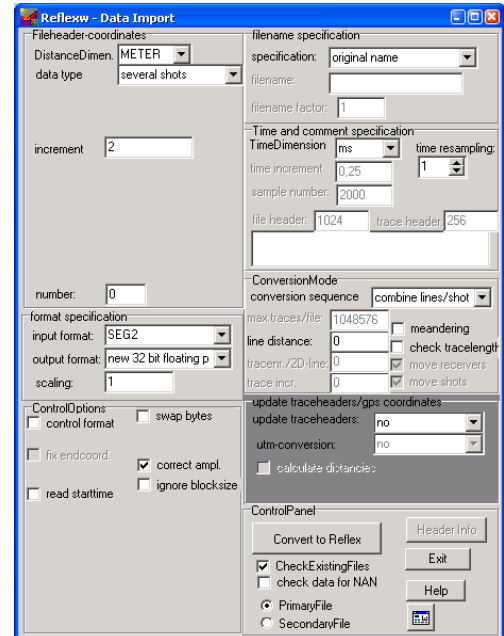
layout). Use this option for example to define the geometry of a borehole containing the shots and the receivers placed at the surface.Shotoffset specifies the x-position of the shot borehole and receiver offset specifies the location of the receivers in y-direction (normally 0 for surface).
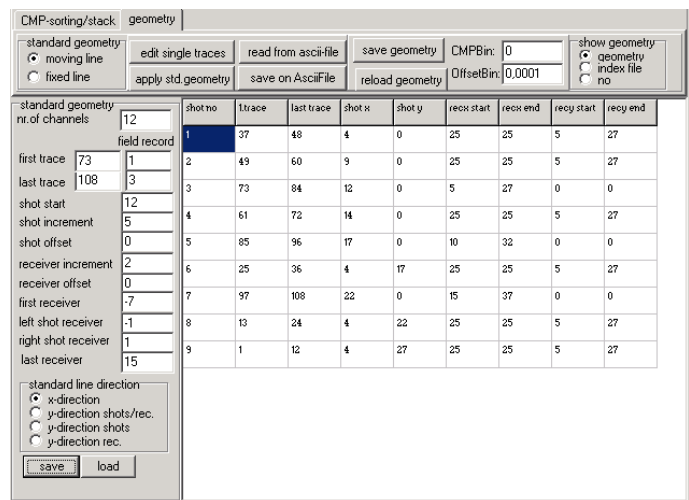
y-direction rec. activated: the receivers are assumed to be orientated in y-direction (borehole/surface layout). Use this option for example to define the geometry of a borehole containing the receivers and the shots placed at the surface. Receiver offset specifies the x-position of the receiver borehole and shot offset specifies the location of the shots in y-direction (normally 0 for surface).

Click on apply std. geometry - the geometry will be updated and save the geometry. It is recommended to check the geometry of the individual traces using the edit single traces.

The standard geometry must be applied separately on each individual configuration. A new configuration is given for example if the receiver line has been changed or if the shots and receivers have been exchanged. The parameters first trace and last trace define the range for the individual configuration.

The following example is a dataset containing three different configurations corresponding to the three layouts of chap. IV.1.1.
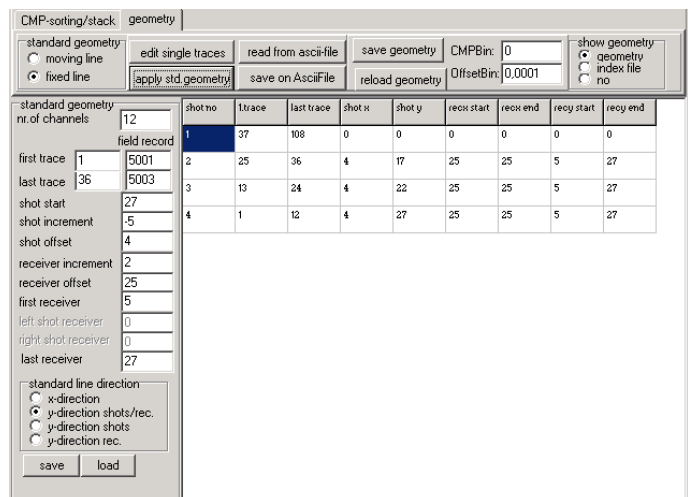
The parameters for the **surface/surface** configuration (layout 1 of chap. IV.1.1 with 12 receivers) are the following:

nr. channels: 12
first trace: 73
last trace: 108
shot start: 12
shot increment: 5
shot offset: 0
receiver increment: 2
receiver offset: 0
First receiver: -7
left shot receiver: -1
right shot receiver: 1
last receiver: 15
x-direction shots/rec. activated

| shot no	t.trace	last trace	shot x	shot y	recx start	recx end	recy start	recy end
1	37	48	4	0	25	25	5	27
2	49	60	9	0	25	25	5	27
3	73	84	12	0	5	27	0	0
4	61	72	14	0	25	25	5	27
5	85	96	17	0	10	32	0	0
6	25	36	4	17	25	25	5	27
7	97	108	22	0	15	37	0	0
8	13	24	4	22	25	25	5	27
9	1	12	4	27	25	25	5	27

The parameters for the **borehole/borehole** configuration (layout 2 of chap. IV.1.1 with 12 receivers) are the following:

nr. channels: 12
first trace: 1
last trace: 36
shot start: 27
shot increment: -5
shot offset: 4
receiver increment: 2
receiver offset: 25
First receiver: 5
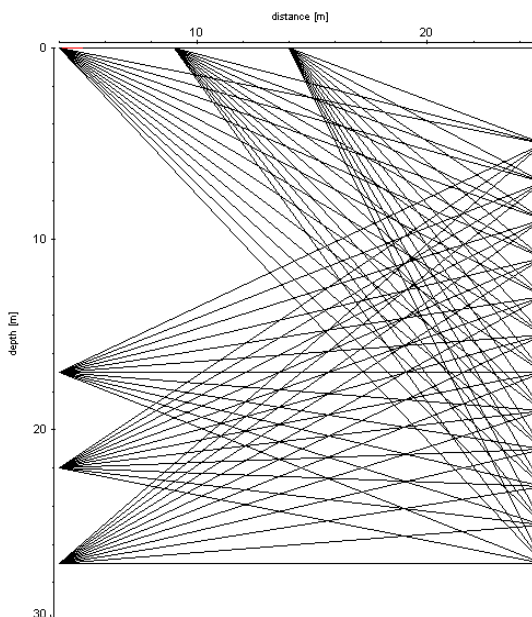last receiver: 27
y-direction shots/rec. activated

shot no	t.trace	last trace	shot x	shot y	recx start	recx end	recy start	recy end
1	37	108	0	0	0	0	0	0
2	25	36	4	17	25	25	5	27
3	13	24	4	22	25	25	5	27
4	1	12	4	27	25	25	5	27

The parameters for the **borehole/surface** configuration (layout 3 of chap. IV.1.1 with 12 receivers within the borehole) are the following:

nr. channels: 12
first trace: 37
last trace: 72
shot start: 4
shot increment: 5
shot offset: 0
receiver increment: 2
receiver offset: 25
First receiver: 5
last receiver: 27
y-direction rec. activated
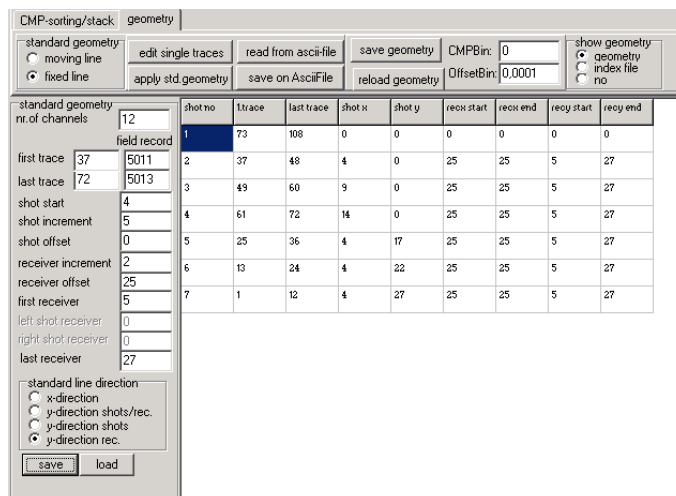
The rays of the two configurations layout 1 and layout 2 are shown on the right (option show rays within the modelling menu).

The geometry of layout 1 cannot be controlled using the show ray option as source and receivers are placed at the surface. A control can only be done using the edit trace header option or using the option check rays within the modelling menu and a manual control of the source/receiver coordinates.

IV.1.3 picking the first arrivals

After having imported the data and having defined the traceheader geometry the traveltime data must be picked.

For that purpose activate the option **pick** and pick the data using one of the picking options. Open the pick save menu using the option **save**. The save picks menu opens (see figure on the right). In any case the picks also should be saved using the Reflex Win format in order to have the possibility to load them again in a later stage.

Use the format ASCII-2D tomography or ASCII-3D tomography in order to generate the ASCII-file for a subsequent tomography. With the option "export several existing picks into 1 ASCII-file" activated you may export several existing pickfiles into 1 ASCII-file. The pick-file will have the extension TOM and will be stored under the path ASCII under the current projectpath.
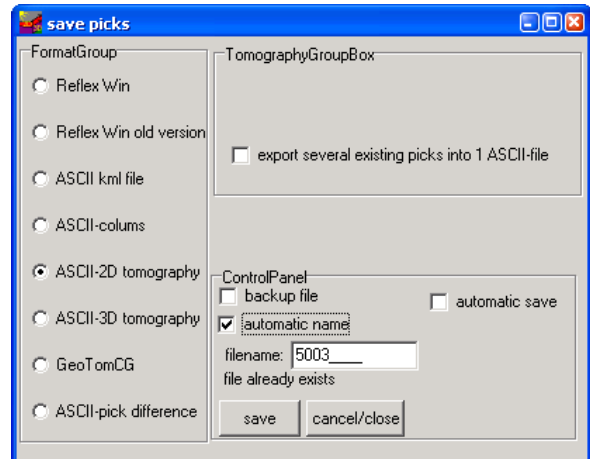
IV.2 performing the transmission tomography

1. First a starting model must be generated (see chap. I) or an already existing model must be loaded using the option file/load model. Normally the starting model may be a simple homogeneous model whereby the velocity should be within the expected range.

2. Activate the option **Tomo**

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

- Load the data using the option **load data** (see also item 1). If the 3D-data format is used for the 2D-tomography you have to deactivate the option **use 2D-data** and you have to specify the **second coordinate** (y or z) within the radiobox **sec.coord.** The first coordinate is always x. The third coordinate is neglected.
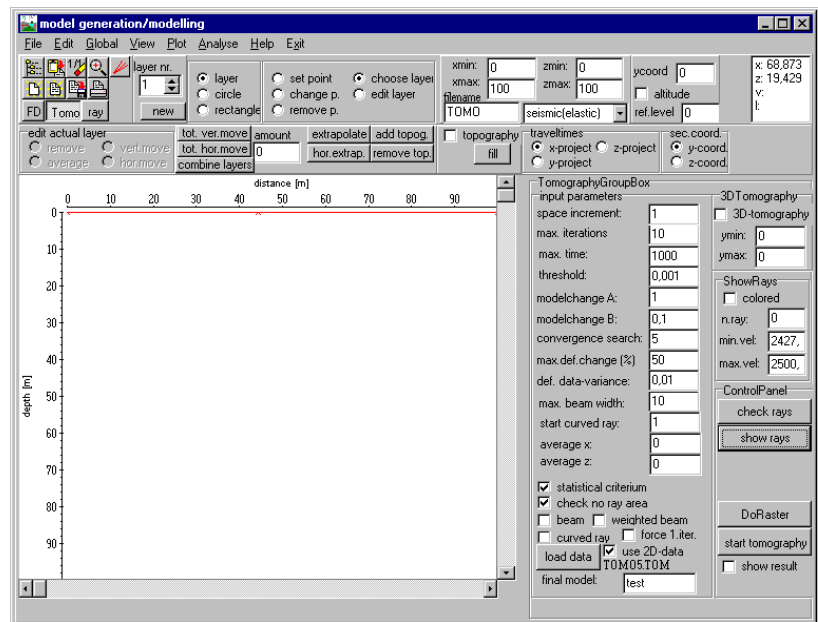
- Check the geometry of your loaded traveltimedata using the option **show rays**

- Enter the wanted **space increment** (equal in x- and z-direction). This increment should be within the range of the receiver or shot increment.

- Activate the option **curved ray** if the curved raytracing shall be used. If activated the option **start curved ray** specifies the iteration step for which the curved raytracing will be used first.

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

- Enter a name for the final model. Please use not the same name as for the starting model because this may lead to problems.

- **Start** the tomography. The tomographic result is stored using the "normal" REFLEXW format. You may display the result within the 2D-dataanalysis.
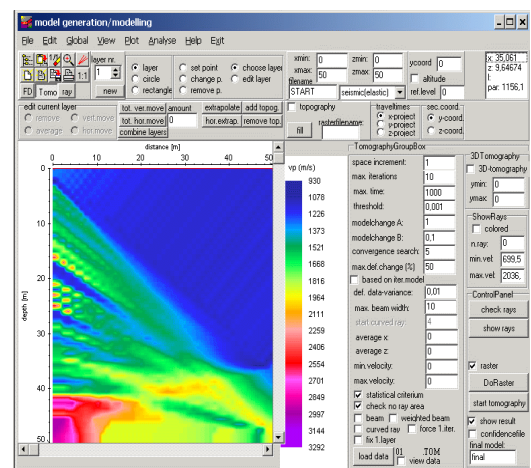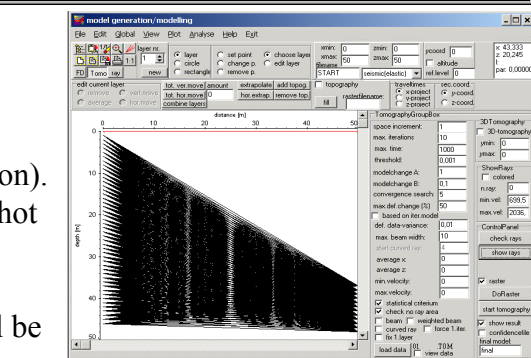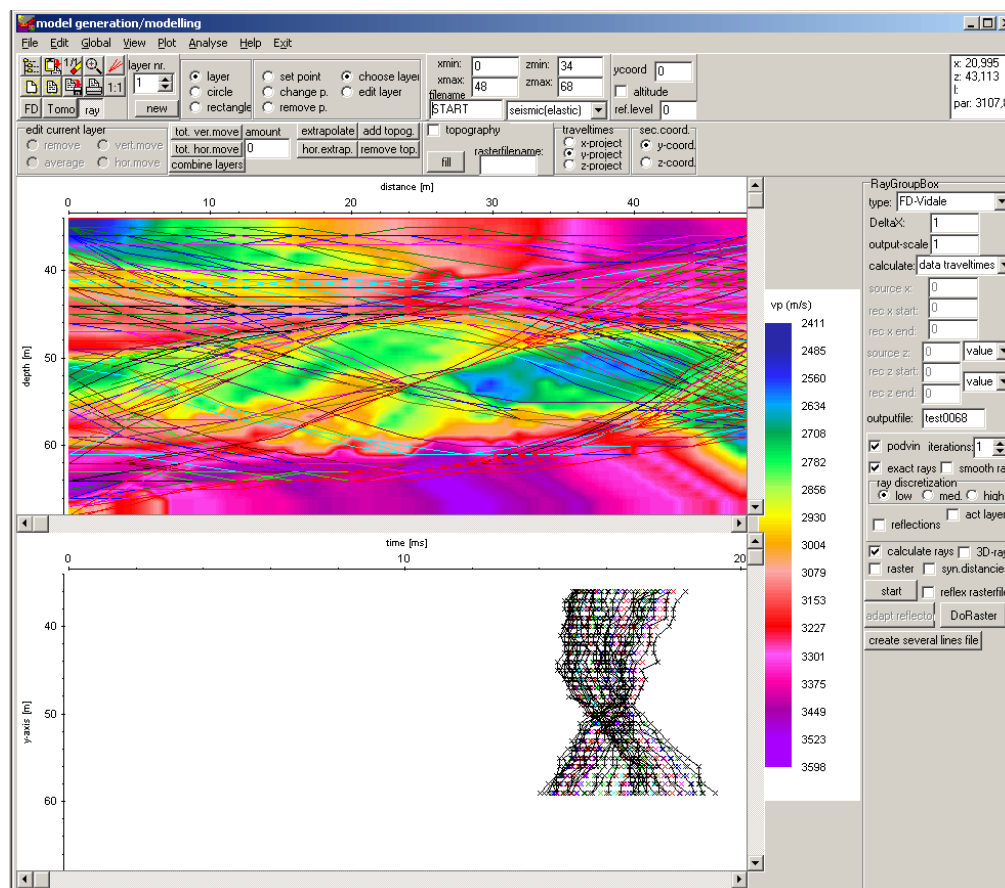
4. Control the tomographic result by a forward raytracing. For that purpose activate the option **ray**. The raytracing menu opens in addition.

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. Activbate the option y-project for traveltimes and y-coord. for sec.coord if the second coordinates have been stored on the y-coordinates and z-project for traveltimes and z-coord. For sec.coord if the second coordinates have been stored on the z-coordinates respectively.

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 tomography.
- 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 raster file. Choose the tomography raster file.
- 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**

Sandmeier geophysical software - REFLEXW guide

IV.3 performing the refraction tomography

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 (see raytracing). 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 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 (see chap. I) or an already existing model must be loaded using the option file/load model. Normally the starting model may be a simple homogeneous model with a quite strong vertical velocity gradient (dv/dz = 50 1/m, e.g.) whereby the velocity at the surface boundary should be within the expected range (e.g. v=400 m/s).

2. Activate the option **Tomo**

3. The **TomographyGroupBox** opens in addition (see figure on the right). Within this group box you have to enter the necessary tomography parameters.
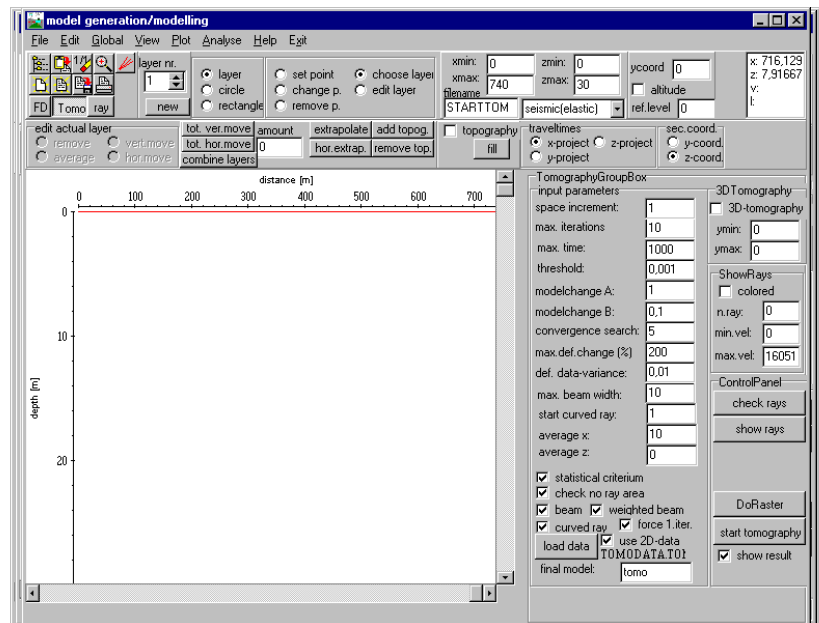
- Load the data using the option **load data** (see also item 1). If the 3D-data format is used for the 2D-tomography you have to deactivate the option **use 2D-data** and you have to specify the **second coordinate** (y or z) within the radiobox **sec.coord.** The first coordinate is always x. The third coordinate is neglected.

- Enter the wanted **space increment** (equal in x- and z-direction). Normally this increment should be small enough in order to allow small scale variations with depth. It may 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 (%)**, e.g. 200 %

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

- Enter a smoothing value in x-direction (parameter **average x**, e.g. 10).

Sandmeier geophysical software - REFLEXW guide

- 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 but 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.

 4. Control the tomographic result by a forward raytracing. 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 tomography.
- 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 raster file. Choose the tomography raster file.
- 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**