

REFLEXW

32 and 64 bit

Version 10.4

Windows™ 7/8/10/11-program
for the processing of
seismic, acoustic or electromagnetic
reflection, refraction and transmission data

Copyright 1998-2024 by

Dr. K.J. Sandmeier
Zipser Straße 1
D-76227 Karlsruhe
Germany
Tel. (49)721/491206

Please feel free to contact us.
e-mail: info@sandmeier-geo.de
Web: www.sandmeier-geo.de

All companies and product names are trademarks or registered trademarks of their respective holders.

Business and Licence conditions for purchasing the Licence of the Program package REFLEXW

By purchasing the licence of the program package REFLEXW the user has accepted the following utilization, warranty and liability limitations.

§1. COPYRIGHT

The program REFLEXW and the documentation are protected by the copyright law of June 26, 1985 and Copyright 1993-2024 by Karl-Josef Sandmeier. Therefore it must not be reproduced, distributed, let, hired out or resold without permission in writing from K.-J. Sandmeier. Only for the purpose of saving and archiving a copy on disc may be produced. All rights for this program and the manual are reserved by K.J. Sandmeier, Zipser Straße 1, 76227 Karlsruhe. REFLEXW is written with DelphiXE2/Delphi11 created using Borland International 2011,2021.

§2. LICENSE CONCESSION

The program package REFLEXW is licensed, not sold. The legal use of the program and the documentation entitles to the usage of the REFLEXW-program each on one data processing device (computer) within the purchasing institute or company location.

A site licence entitles to the installation and use of the program REFLEXW on an unlimited number of data processing devices within the purchasing institute or company location.

A testlicence of REFLEXW is restricted to non commercial use and ends after the fixed test period. The total program must be returned together with the manual and the hardware key if existing and all existing copies must be removed.

§3. WARRANTY

1. REFLEXW is an extensive program package for the recording and processing of reflection and transmission data. No warranty is made for the correctness of the program package REFLEXW and for the correctness and completeness of the documentation. However, it is guaranteed to the user, that the program package is useable in the sense of the program documentation that is valid at the time of delivery to the user and possesses the features guaranteed there.

2. K.-J. Sandmeier guarantees that the original program package is duly recorded on a certified data storage device.

3. If a program package proves not useable as defined by paragraph 1 or defective within the meaning of paragraph 2, within a warranty period of six months, beginning with the delivery of the program package to the user, the delivered program package is taken back and exchanged free of charge for a new program package with the same title. In case that this one also proves unuseable within the meaning of paragraph 1 or defective as defined by paragraph 2 and K.-J. Sandmeier is unable to establish applicability with a reasonable effort in a reasonable amount of time, the user has according to his choice the right of a reduction of the purchase price or to return the program package and receive a refund of the purchase price.

4. Further warranty obligations do not exist. Especially no warranty is made that the program package complies with the special demands of the user. The user alone is responsible for selecting, installing and using and also for the intended results.

§4. LIABILITY

1. The applicability of the program package for a special purpose is not guaranteed. Also no liability is taken in any way for incidental or consequential damages arising from the use of this program package (these are among others without limitation damages due to losses of business profits, operating troubles, loss of business data or other financial losses). This exclusion of liability equally applies to damages to a third party.

§5. The court of jurisdiction is Karlsruhe, Germany

Table of Contents

About REFLEXW	12
0. REFLEXW user's guide	23
0.0 the different REFLEXW modules	23
0.1 Directory structure/project directory	25
0.2 some general comments	28
0.3 Import and first display of GPR-data	31
0.3.1 Import GPR-data	31
0.3.2 Display the data	32
0.4 Filtering the imported data	34
0.4.1 General overview of filtering	34
0.4.2 Filtering of a single datafile	34
0.4.3 Filtering different 2D-lines using the same filter sequence (sequence processing)	38
0.4.4 Handling of a strong topography for ZeroOffset data	40
0.5 Picking within REFLEXW	47
0.5.1 Picking single objects	47
0.5.1.1 Picking hyperbola along one profile line	47
0.5.1.2 Use of xy-coordinates defined within the fileheader	48
0.5.1.3 control and output of single picks for several profiles	50
0.5.1.4 Picking linear structures within a 3D-datafile	51
0.5.2 Picking different reflectors within REFLEXW	53
0.5.2.1 General flow	53
0.5.2.2 pick a distinct reflector	53
0.5.2.3 generate a velocity file for the time-depth conversion (open layershow panel)	54
0.5.2.4 combine the different picked reflectors into one layer model (open layershow panel)	56
0.5.2.5 generate an output report of the layer-show (open layershow panel)	58
0.6 coordinates	59
0.6.1 peculiarities for meandering 2D-profiles	59
0.6.2 Use of GPS-coordinates	60
0.6.2.1 Set the GPS-coordinates	60
0.6.2.1 Handle the traceheader(GPS)-coordinates	61
0.6.2.1.1 picking	61
0.6.2.1.2 viewing options within the 2D-dataanalysis	61
0.6.2.1.3 processing options	62
0.6.2.1.4 3D-datainterpretation	62
0.7 Introduction to seismic refraction data interpretation	63
0.7.1 Import the data and pick the first onsets (done within the module 2D-dataanalysis)	63
0.7.2 interpretation of the picked traveltimes (done within the module travelttime analysis 2D)	67
0.7.3 Forward raytracing to refine the model (done within the module model	

generation/ modelling)	71
0.7.4. Refraction-tomography: A second kind of interpreting the data . .	73
0.7.5. Joint interpretation of the results of the Wavefront-Inversion and the Refraction-tomography	77
0.7.6 Topography	79
0.8 Introduction to the processing of seismic reflection data within REFLEXW	85
0.8.1 Import the data (done within the module 2D-dataanalysis)	85
0.8.2. Crosscorrelation for vibration data (done within the module 2D- dataanalysis)	88
0.8.3. setting the geometry.	90
0.8.4 pre-stack filtering (done within the module 2D-dataanalysis). . .	92
0.8.4.1 prestack filter sequence for data exhibiting surface waves	93
0.8.4.2 prestack filter sequence for horizontal vibration data	97
0.8.5 velocity analysis and stacking (done within the module 2D- dataanalysis)	99
0.8.6 Prestack migration as an alternative to the standard NMO-stacking	103
0.9 Introduction to the 3D-datainterpretation within REFLEXW	104
0.9.1. generation of a 3D-dataset.	104
0.9.1.1 Generating a 3D-file without interpolation	106
0.9.1.1.1 Generating a 3D-file from during the import from parallel 2D-lines	106
0.9.1.1.2 Generating a 3D-file from Reflexw formatted parallel 2D- lines	109
0.9.1.2 Generating a 3D-file with interpolation (done within the 3D- datainterpretation).	111
0.9.1.2.1 Generating a 3D-file from parallel or crossing 2D-lines	111
0.9.1.2.2 Generating a 3D-file for freely distributed 2D-lines .	113
0.9.1.3 resulting REFLEXW 3D-datafile	115
0.9.1.4 combine different 3D-datafiles.	116
0.9.2 processing a 3D-data file (done within the 2D-dataanalysis) . . .	117
0.9.3 interpretation of a 3D-data file (done within the 3D-datainterpretation)	118
0.9.3.1 load a 3D-datafile	118
0.9.3.2 display a 3D-datafile using the scroll option	118
0.9.3.2.1. xyz-cuts option	119
0.9.3.3 display a 3D-datafile using the windows option.	120
0.9.3.4 display a 3D-datafile using the 3D-cube option.	121
0.9.4 3D-view of single 2D-lines with arbitrary geometry	122
0.10 Introduction to the modelling/tomography tools within REFLEXW . .	124
0.10.1 model generation	124
0.10.1.1 create a new model	124
0.10.1.2 change an existing model	126
0.10.2 Finite Difference (FD) modelling	127
0.10.3 ray tracing	129
0.10.4. tomography	131
0.10.4.1 picking the travelttime data and description of the format	131
0.10.4.1.1 single shot analysis	132

Table of Contents

0.10.4.1.2 several shots analysis	138
0.10.4.1.3 picking the first arrivals	142
0.10.4.2 performing the transmission tomography	143
0.10.4.3 performing the refraction tomography	146
0.11 Use of CMP-analysis for a single or several independent CMP-files	149
0.11.1. Import the data data for single CMP	150
0.11.2. Processing single GPR CMP	151
0.11.3. velocity analysis	152
0.12 Introduction to the use of 2D-velocity models for further processing	154
0.12.1. 2D-model from the interactive diffraction analysis	155
0.12.2. 2D-model from CMP velocity analysis	156
0.12.3. 2D-model from the layershow (picked interfaces)	158
0.12.4. 2D-model from the modelling menu, e.g. from refraction data	159
0.12.5. 2D-model from tomography	161
0.13 GPS based Radar array data	162
0.13.1. Import the data	165
0.13.2. Mira from Mala	167
0.13.3. IDS Stream C	169
0.13.4. ImpulseRadar (Raptor)	173
0.13.5. Utsi	179
0.13.6. 3D-Radar	180
0.14 general considerations	182
0.14.1 Which data types can be imported?	182
0.14.2 The difference between FileHeader and Traceheader	182
0.14.3 How is the FileHeader set or changed?	183
0.14.4 How is the TraceHeader set or changed?	183
0.14.5 How are the PlotOptions set or changed?	183
0.14.6 Some special recommendations about geometry and preprocessing	184
0.14.6.1 Fileheader	184
0.14.6.2 Coordinates	184
0.14.6.3 Shot time	184
0.14.6.4 Peculiarities	185
0.14.7 Getting information about the velocity of the medium	186
0.14.7.1 Migration	186
0.14.7.2 ZO-data / CO-data	186
0.14.7.3 CMP-Measurements / Moveout-Measurements	186
0.14.7.4 Using Coredata to get velocity information	187
0.14.7.5 Generating 2D-velocity models	188
0.14.8 Picking of onsets	189
0.14.8.1 Preconditions for picking	189
0.14.8.2 Setting of picks	189
0.14.8.3 Saving of picks	189
0.14.8.4 Saving of picks to create a layershow	190
0.14.8.5 Export of picks to create contour plots	190
0.14.8.6 Picking and time-depth conversion of several layer boundaries	191
1. 2D-Data-Analysis	193
1.1 File Menultem	201
1.1.1 interactive choice	203

Table of Contents

1.2 Global MenuItem	208
1.2.1 Global settings	208
1.3 Plot MenuItem	213
1.4 View MenuItem	214
1.5 Import Menu	225
1.5.1 Import	238
1.5.2 Import filename specification	241
1.5.3 Import ControlOptions	243
1.5.4 Import format specification	250
1.5.5 Import ConversionMode	253
1.5.6 Import ControlPanel	258
1.5.7 Import time/comment specification	261
1.6 Export Menu	262
1.7 PlotOptions	266
1.7.1 PlotSettings	266
1.7.2 Pointmodeattributes	270
1.7.2.1 edit color palettes	273
1.7.3 Wiggleattributes	275
1.7.4 Manual scaling and incrementation	276
1.7.5 PlotGain/Filter	276
1.7.6 Plotsuboptions	277
1.7.7 ControlPanel	281
1.8 Print Menu	282
1.8.1 PrintOptions1	282
1.8.2 PrinterSize	283
1.8.3 PrintOptions2	283
1.8.4 FontSettings	284
1.8.5 ControlPanel	285
1.8.6 Printing on banner paper	285
1.8.7 Print preview	286
1.9 FileHeader Edit	288
1.9.1 Fileheader coordinates	288
1.9.2 Fileheader time specification	289
1.9.3 Fileheader input1	289
1.9.4 Fileheader filename specification	291
1.9.5 Fileheader ControlPanel	292
1.9.6 Traceheader Edit	293
1.9.6.1 traceheader tabella	316
1.10 Edit several FileHeaders	324
1.11 Processing MenuItem	330
1.11.1 1D-Filter	331
1.11.1.1 Meanfilter	333
1.11.1.2 Medianfilter	333
1.11.1.3 Bandpassfrequency	334
1.11.1.4 Bandpassbutterworth	335
1.11.1.5 Filter/timedependent	336
1.11.1.6 Notchfilter/frequ.	337
1.11.1.7 spectrum spikes	337
1.11.1.8 Subtract-mean (dewow)	338
1.11.1.9 Subtract-DC-shift	338

1.11.1.10 Crosscorrelation	338
1.11.1.11 auto correlation	339
1.11.1.12 Resampling	339
1.11.1.13 Walsh bandpass	340
1.11.2 Gain	341
1.11.2.1 AGC-Gain	343
1.11.2.2 Energy decay	343
1.11.2.3 Gain function	344
1.11.2.4 Remove header gain	344
1.11.2.5 div. compensation	345
1.11.2.6 Manual gain (y)	345
1.11.2.7 Manual gain (x)	346
1.11.2.8 Scaled windowgain(x)	347
1.11.2.9 trace normalize	347
1.11.2.10 x-distance decay(db)	348
1.11.2.11 compensate stripes	348
1.11.2.12 normalize profiles	349
1.11.2.13 normalize 3D-file	350
1.11.3 StaticCorrection/muting	351
1.11.3.1 Static correction	353
1.11.3.2 Dynamic correction	355
1.11.3.3 Move starttime	355
1.11.3.4 Muting	356
1.11.3.5 surgical muting	357
1.11.3.6 Time cut	358
1.11.3.7 Correct max. phase	359
1.11.3.8 Corr.max. phase/wrap	360
1.11.3.9 Correct picked phase	361
1.11.3.10 Correct for 2 layers	362
1.11.3.11 Correct 3D-topography	363
1.11.3.12 Suppress multiples	367
1.11.4 Declipping/arithmetic functions	368
1.11.4.1 Declipping	370
1.11.4.2 Unwrapping	371
1.11.4.3 Arithmetic functions	372
1.11.5 Complex trace-analysis/spectral analysis	373
1.11.5.1 Complex trace-analysis	375
1.11.5.2 Trace spectrum	375
1.11.5.3 Moving window spectrum	376
1.11.5.4 Rotate 2 components	377
1.11.5.5 Add random noise	377
1.11.5.6 Trace phase-spectrum	378
1.11.5.7 texture attribute	378
1.11.6 Trace Interpolation/Resorting	380
1.11.6.1 Markerinterpol	381
1.11.6.2 Traceincr-resampling	384
1.11.6.3 Make equidist.traces	384
1.11.6.4 fix profile length	385
1.11.6.5 split file	385
1.11.6.6 YFlipProfile	386
1.11.6.7 XFlipProfile	386
1.11.6.8 Resort-traceheader	386

1.11.6.9 Resort-group	387
1.11.6.10 XY-exchange	387
1.11.6.11 Yo-Yo section	388
1.11.6.12 VSP corridor stack	390
1.11.6.13 Traceinterpol-3DFile	391
1.11.6.14 XFlip-3DFile	391
1.11.6.15 Shift-3DFile	391
1.11.6.16 create 3D-ensembles	392
1.11.7 Edit traces/traceranges	393
1.11.7.1 Remove	395
1.11.7.2 Remove range	395
1.11.7.3 Interpolate	396
1.11.7.4 Extract	397
1.11.7.5 Replace	398
1.11.7.6 Reverse polarity	398
1.11.7.7 Set to zero	399
1.11.7.8 Duplicate	399
1.11.7.9 Move	400
1.11.7.10 Swap 2 timeblocks	400
1.11.7.11 Insert profile	400
1.11.7.12 Combine files f.CMP	401
1.11.7.13 merge files	402
1.11.7.14 merge in timedir	404
1.11.7.15 insert zero traces	404
1.11.7.16 remove zero traces	405
1.11.7.17 remove idle traces	405
1.11.8 2D-Filter	406
1.11.8.1 Running average	408
1.11.8.2 Subtracting average	408
1.11.8.3 Background removal	409
1.11.8.4 Compress	410
1.11.8.5 Expand	410
1.11.8.6 Stack traces	411
1.11.8.7 Subtract traces	411
1.11.8.8 Compress 3D-file	412
1.11.8.9 Expand 3D-file	412
1.11.8.10 average xy-filter	412
1.11.8.11 median xy-filter	413
1.11.8.12 dilation xy-filter	413
1.11.8.13 erosion xy-filter	413
1.11.8.14 constrain borders	414
1.11.8.15 average 3D-filter	414
1.11.9 Migration	415
1.11.9.1 Diffraction stack	417
1.11.9.2 Kirchhoff migration	419
1.11.9.3 Fk migration (Stolt)	421
1.11.9.4 topography migration	422
1.11.9.5 diffraction 2D-veloc	426
1.11.9.6 Kirchhoff 2D-veloc	427
1.11.9.7 FD-migration	428
1.11.9.8 timedepth conversion	429
1.11.9.9 semblance	430

1.11.9.10 prestack migration	431
1.11.9.11 3D-diffraction stack	434
1.11.9.12 3D-Kirchh. migration	435
1.11.9.13 3D-fk migration	436
1.11.9.14 3D-Kirchh. 2D-vel.	437
1.11.9.15 3D-semblance	438
1.11.10 fk-filter/data fusion	439
1.11.10.1 Fk filter.	441
1.11.10.2 Fk filter-lineparts	443
1.11.10.3 Fk spectrum.	444
1.11.10.4 dispersion curve	445
1.11.10.5 data fusion.	446
1.11.10.6 Mix profile	449
1.11.11 Deconvolution/shaping	450
1.11.11.1 Deconvolution	451
1.11.11.2 Deconvolution/shap.	451
1.11.11.3 Spectral whitening	453
1.11.11.4 differentiation.	454
1.11.11.5 Integration	454
1.11.11.6 Shaping	455
1.11.11.7 extract wavelet.	456
1.11.12 Sequence Processing	457
1.12 Analysis Menultem	461
1.12.1 Velocity adaptation	462
1.12.2 Pick Menultem.	466
1.12.2.1 Pick save Menultem	480
1.12.2.2 Auto-Pick Menultem	487
1.12.3 LayerShow Menultem	491
1.12.3.1 Create LayerShow.	494
1.12.3.2 Create LayerShow report.	497
1.12.3.3 Create LayerShow velocities.	501
1.12.3.4 LayerShow amplitude calibration.	504
1.12.3.5 LayerShow multioffset picks	506
1.12.4 CMP-processing	508
1.12.4.1 CMP-processing geometry	511
1.12.4.2 CMP-processing stack.	517
1.12.5 edit comment markers	521
1.12.6 adjust trace gain	525
1.12.7 3-component analysis	526
1.12.8 create mpeg file	529
1.12.9 2D-lines statistics	530
2. CMP(1D) velocity analysis.	532
2.1 CMP velocity adaptation	542
2.2 CMP semblance/Const.velocity analysis	546
2.3 CMP 2D-model	549
2.3.1 2D vrms velocities.	549
2.3.2 2D interval layer velocities.	550
2.4 CMP practical aspects	551
2.5 calculation of synthetic seismograms	552

3. 3D-datainterpretation	555
3.1 3D-datainterpretation Menuitems	559
3.1.1 3D-File Menuitem	559
3.1.2 Scan 3D.	562
3.1.3 3D-fixed array	562
3.1.4 View Menuitem	563
3.1.5 Analyse Menuitem.	568
3.1.6 3D-file processing	571
3.2 Scroll 3D-file	573
3.2.1 xyz-cuts	580
3.3 Windowing 3D-file.	583
3.4 3D-cube display	588
3.5 Generate 3D-file from 2D-lines	593
3.6 Generate single timeslices	598
3.6.1 Timeslice calculation example.	601
3.6.2 Timeslice calculation overview	603
3.7 view fast data cube display	605
3.8 3D picking	607
3.8.1 Picking a crooked target object with depth variations	612
3.9 3D-fixed array	614
3.9.1 3D-fixed array Menuitems	616
3.9.2 3D-fixed array timeslices	619
3.9.3 3D-fixed array velocity adaptation	620
3.9.4 3D-fixed array picking	621
3.9.4.1 2D-picking	621
3.9.4.2 3D-picking	621
3.9.5 3D-fixed array layer show	623
3.9.6 3D-fixed array gps map	624
3.10 Scan 3D	627
4. Modelling	637
4.1 Modelling File Menuitem	644
4.2 Modelling Edit Menuitem.	647
4.3 Modelling View Menuitem	649
4.4 Model generation	651
4.4.1 Getting started.	651
4.4.2 Generating a new layer	651
4.4.3 Input of model parameters window	652
4.4.4 Changing an existing layer	657
4.4.5 Topography	659
4.5 FD-Model simulation	660
4.5.1 FD-computation.	660
4.5.2 Computation of several single lines	666
4.5.3 Absorbing Range for the e.-m. wave propagation.	667
4.5.4 FD-modelling 3D	668
4.6 Tomography	669
4.7 raytracing/wavefront inversion	677
4.7.1. raytracing 2D	678
4.7.1.1 raytracing shot/receivers	680
4.7.1.2 raytracing reflections	682
4.7.1.3 automatic adaptation of primary reflections.	684
4.7.2 wavefront-inversion 2D	687

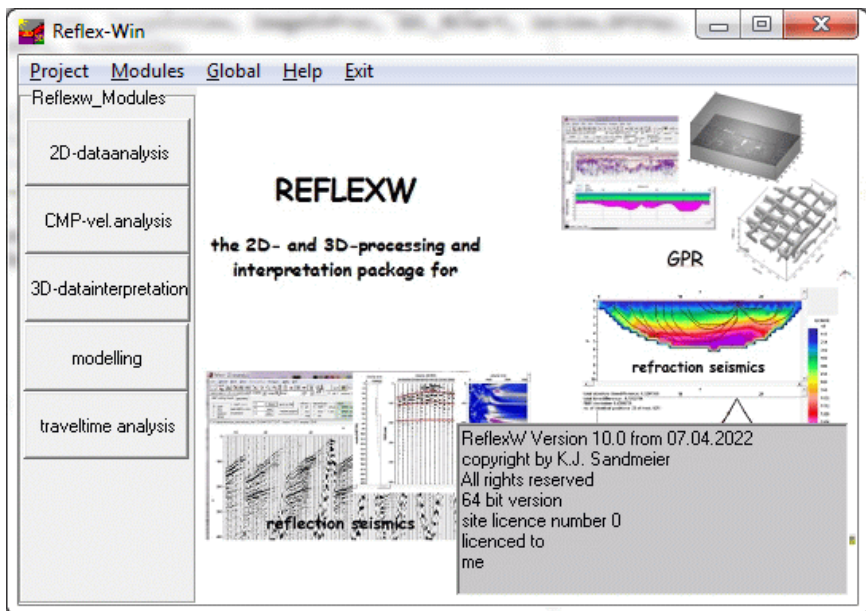
5. Refraction traveltime analysis 2D	688
5.1 traveltimes put together	691
5.2 traveltimes assign	695
5.3 traveltimes combine	697
5.3.1 combine traveltimes for the uppermost layer	697
5.3.2 combine traveltimes for deeper layers	697
5.4 time term analysis	700
6. Seismic crosshole testing	702
6.1. Original data.	702
6.2. Channel sorting and filtering within seismic crosshole testing	703
6.3. Apply borehole deviations and edit geometry	704
6.4. Arrival time picking	708
Literature	712

About REFLEXW

Short program description:

REFLEXW is a Windows™9x/NT/XP/Vista/7/8/10 program for the processing and interpretation of reflection and transmission data (special applications: ground penetrating radar (GPR), reflection and refraction seismics and ultrasound).

graphics resolution: It is recommended to use true color resolution. The fonts and the glyphs of the buttons will be dynamically adapted to the used scale value of the text. Normally 96 dpi are used for 100% scale. If a larger value will be chosen the glyphs will be automatically adapted.



The following modules are available:

module **2D data-analysis** with

- easy import of data of many different formats (e.g. SEGY, SEG2, most of the EMR- and seismic systems (for example: GSSI, MALA RD3, PULSE-EKKO) and so on), integration of other nonstandard formats.
- conversion of single sections and of a profile sequence (automatic assembling and storing of the sections under one single data file or automatic generation of data files for parallel and inline-sections).
- many different display and processing possibilities
- interactive processing of single files or by generating a batch-file (sequence

- processing)
- one- and multi-channel filters, editing, static correction, deconvolution, migration and much more.
- interactive velocity adaptation
- picking of onsets
- layer-show
- CMP-processing

module **CMP velocity analysis** which the calculation of a one-dimensional velocity-depth-distribution from CMP- or moveout-data based on different analysis techniques with the following possibilities:

- interactive CMP velocity adaptation of a velocity-model for a CMP- or a moveout-section with continuous indication of the current reflections
- CMP semblance analysis for a given velocity-interval, interactive choice of a vrms-depth-distribution from the semblance or unnormalized correlation analysis, direct reconstruction of the interval velocities, possibility to change the resulting distribution by the model generation module described above
- generation of a CMP 2D-model based on the resulting 1D-velocity-depth distributions. This 2D-model represents the base for the stacking, the migration and the depth-conversion.

module **3D data-interpretation** for displaying x-,y- or z-slices with the following possibilities:

- easy generation of a 3D-dataset from individual 2D-lines which may be parallel, crossing or randomly spaced within the xy-plane
- Display of x-,y- or z-slices. The slices are either displayed in manually scalable windows or by moving through the 3D-dataset using a track bar
- 3D-cube display of the individual slices or of the full 3D-data using shading algorithms
- 3D-Picking of arrivals
- The **scan 3D tool** (see chap. 3.10) allows a fast access to rectangular 3-dimensional GPR- or seismic data which have been acquired along 2D-parallel lines in one or two perpendicular directions without going through the whole process of import, processing and generation.
- The **3D-fixed array tool** (see chap. 3.9) allows a fast interpretation of multichannel data with fixed lateral offset between the channels which allows to easily interactively reach each point within the dataset using the mouse either within the individual profiles or timeslices.

module **modelling** (simulation/inversion) with the following tools:

- interactive generation of a underground model consisting of different layers and different physical parameters for the seismic or electromagnetic wave propagation
- wavefront inversion of refraction traveltimes
- different raytracing methods are available for the evaluation of synthetic traveltimes, e..g. forward modelling of traveltimes using a FD approximation of the Eikonal equation
- forward modelling for simulating the electromagnetic and seismic wave propagation in a 2-dimensional medium using a Finite Difference scheme
- A 2D-tomographic approach based on SIRT is incorporated for the inversion of the picked traveltimes (refraction and transmission). Straight and curved rays are supported. A 3D-tomography is available for transmission traveltimes data.

module **traveltimes analysis 2D** which allows to analyse and interpret picked first arrivals (refraction seismics).

- various sorting and combining possibilities for the picked traveltimes
- these combined traveltimes are the bases for a FD based wavefront inversion

If the main windows will be minimized it is possible to maximize them again using the corresponding button within the main menu.

Installing Reflexw under Windows™Vista, Windows™7 and Windows™10:

There are 2 main differences between running under Vista, 7,8 and 10 and other former Windows systems. :

1. the hlp-files are no longer supported. Therefore the new Reflexw from version 4.5 comes with a chm helpfile.

2. A new User Account Control:

When User Account Control (UAC) is enabled (the default), standard user permissions are used to run applications--even if the user is an administrator. According to Microsoft this "helps eliminate the ability for malware to invoke administrator privileges without a user's knowledge."

Reflexw won't normally crash when it tries to write to a protected resource, such as Program Files. Instead, those writes are virtualized. Trying to write to a file in C:\Program Files\Some Folder causes that file to be created or updated in C:\Users\username\AppData\Local\VirtualStore\Program Files (x86)\Some Folder, where username is the name of the logged-in user.

You must have the rights to write to these protected resources because some files will be stored there. These are:

password.txt - contains your Reflexw access parameters

palette.fil - contains the stored color palettes

reflexw.ini - contains the last Reflexw settings

net.dat - contains the star for the network raytracing

fdem.dat - contains the interface between Reflexw and the fd-simulation program

palette.bmp - is used for a buffer storage

dummy*.bmp - is used for a buffer storage when plotting/printing

dummy*.sta - is used for a buffer storage for the random model generation

TraceHeaderAsciiFormat.txt - contains the free ascii settings for the traceheader

FreeAsciiFormat.txt - contains the free ascii settings for the ASCII picks

zone.txt - is used for the utm conversion

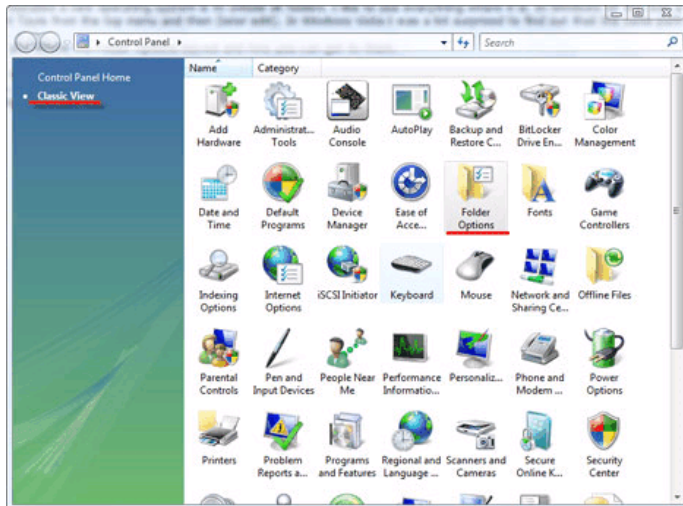
If you do not have the rights for this folder it is recommended to install Reflexw under any path unless C:\Program files (x86)

The folder AppData is **hidden** by default. Therefore the hidden attribute must be eliminated or the hidden files and folders are made visible. For that purpose you have several possibilities. The easiest way to make visible hidden folders is listed within the next two pages.

In order to eliminate the hidden attributes perform the following steps:

Windows 7/8:

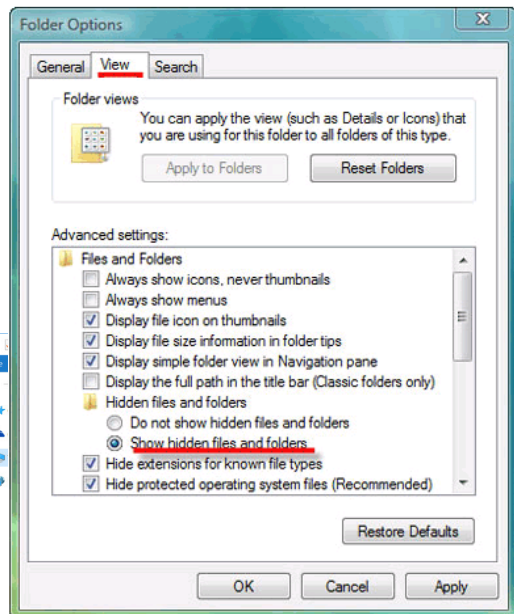
- go to computer and launch the folder C:\Users\username.
- use the classic view within the Control Panel and click on Folder Options.



Within the Folder Options click on View:

- Select (dot) Show hidden files, folders, and drives.
- Click on the Apply button.

Now the hidden directories and files are visible.



Windows 10:

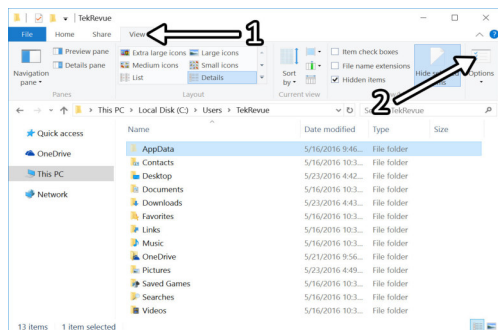
- enter the Explorer and go to computer and launch the

folder C:\Users\username.

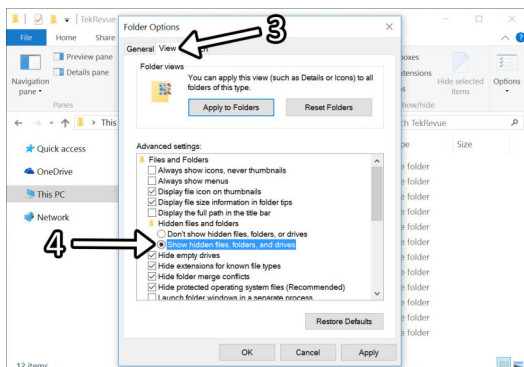
- click on View in the toolbar. A- activate Hidden items
- click on Options and the folder options menu opens.

Within the Folder Options
ck on View:

- in the “Advanced Settings”
t, find and click the radio
utton “Show hidden files,
folders, and drives”.
- Click OK to save your
change and close the Folder
Options window.

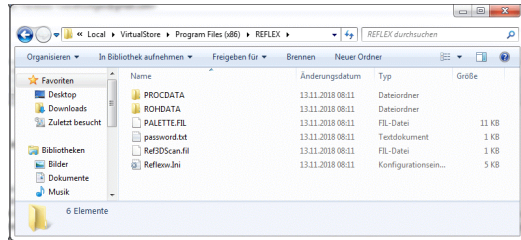


cli
lis
b
fol



The following files will be stored under this “virtual” folder:

password.txt
palette.fil
reflexw.ini
net.dat
fdem.dat
palette.bmp
dummy*.bmp
dummy*.sta
TraceHeaderAsciiFormat.txt
FreeAsciiFormat.txt
zone.txt [utm conversion file)



These files will not be removed when the program Reflexw will be uninstalled. They must be removed manually. The main problem may occur with the password.txt file if you have installed another licence before and you want to use a new licence installed on the same path. In this case you must manually remove the file password.txt before the new installation.

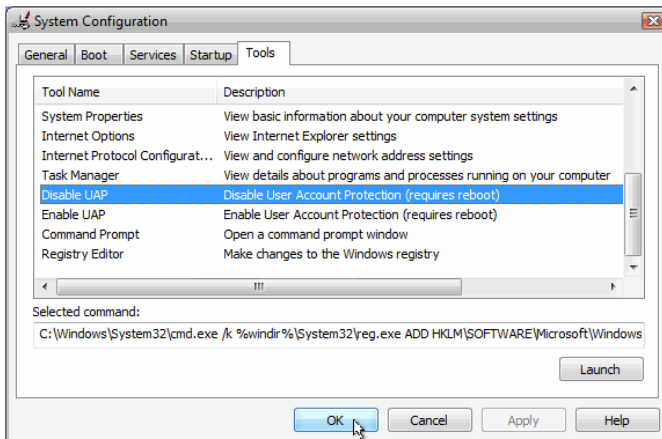
The processed files of the folder demodata will also be written to the virtual folder. This is not visible when running the program Reflexw.

It is strongly recommended not to use any protected folder for the data projects.

If you disable the user account control no difference exists to the former Windows version. Two different method of disabling the user account control are listed below:

Method 1 - Using MSCONFIG

1. Launch MSCONFIG by from the Run menu.
2. Click on the Tools tab. Scroll down till you find "Disable UAP" (this should probably change to UAC in next Vista beta builds and in the RTM version). Click on that line.
3. Press the Launch button.
4. A CMD window will open. When the command is done, you can close the window.
5. Close MSCONFIG. You need to reboot the computer for changes to apply.



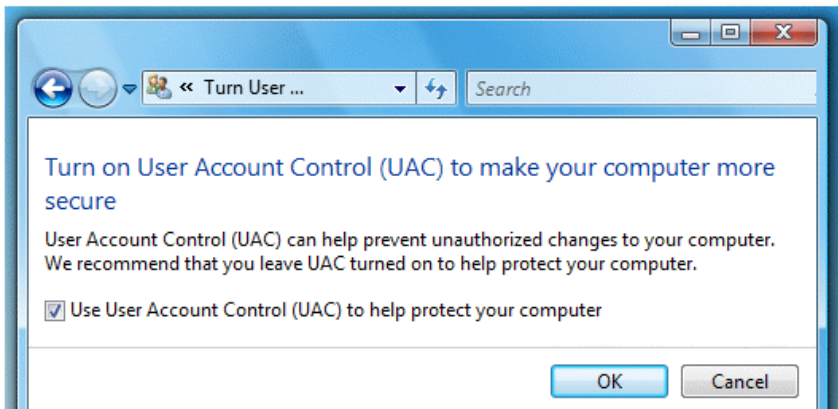
Method 2 - using the user control panel

1. Open Control Panel.
2. Under User Account and Family settings click on "user account"
3. Click on "Turn User Account Control (UAC) on or off" link.



4. In the "Turn on User Account Control (UAC) to make your computer more secure" click to unselect the "Use User Account Control (UAC) to help protect your computer". Click on the Ok button.

5. You will be prompted to reboot your computer. Do so when ready. In order to re-enable UAC just select the above checkbox and reboot.



Windows 10 peculiarities:

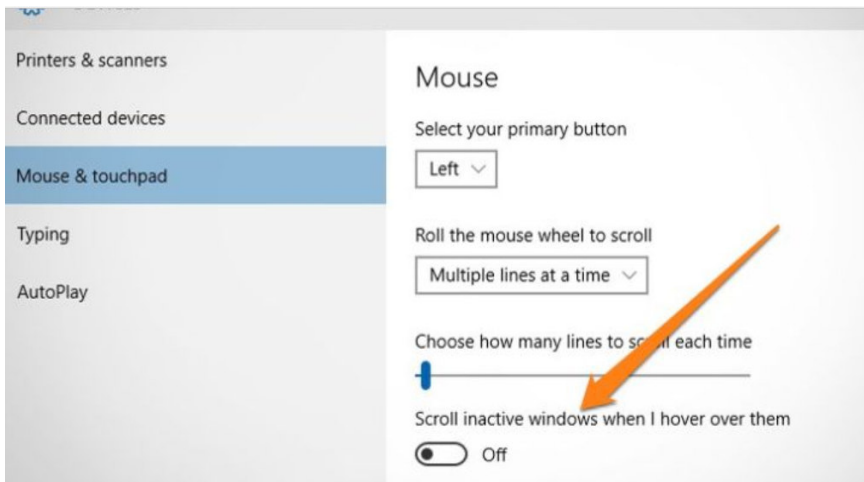
One of the native features of Windows 10 is the ability to scroll inactive windows by just hovering over such a window with the mouse pointer and then using the mouse wheel. If this feature is activated the scroll wheel within Reflexw does not work any more. You must deactivate this feature.

To disable scrolling inactive windows in Windows 10, do the following.

Open the Settings, e.g by the short key Win + I

Go to Devices and then select Mouse

Disable the option Scroll inactive windows when I hover over them



Under a speccal Windows 10 version, namely version 1909 the scroll bars within the 2D-dataanalysis have not been visible. An upgrade to Version 20H2 does not show this problem any more.

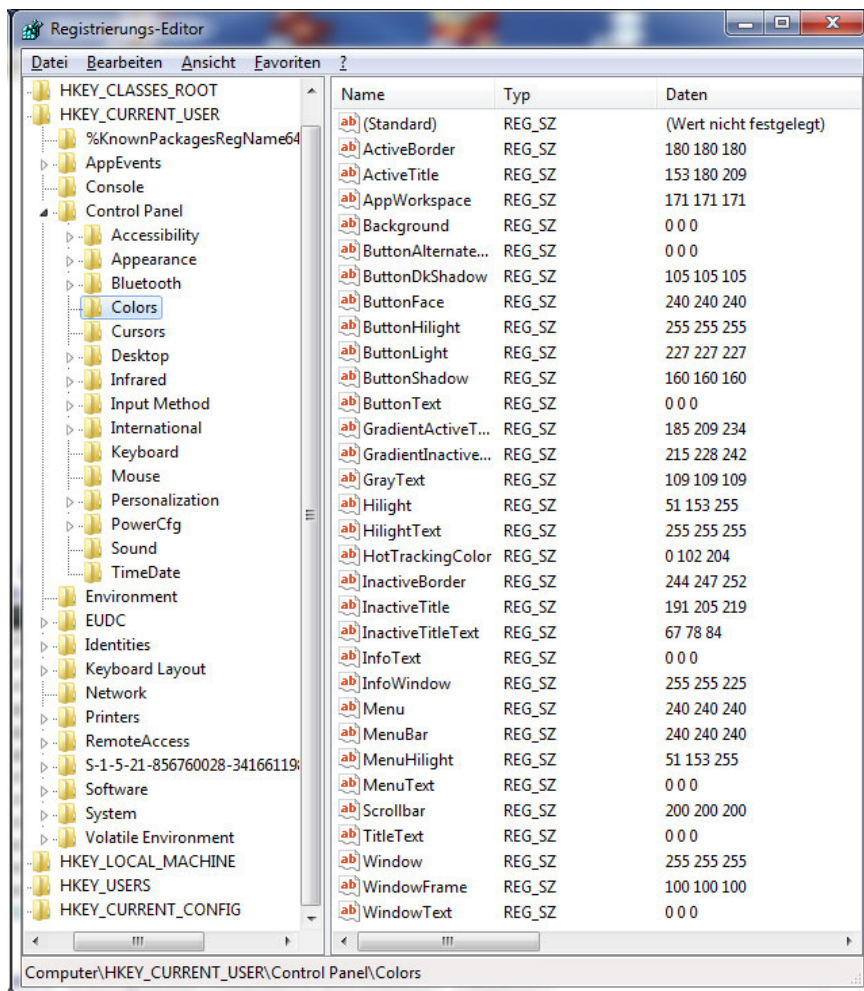
Used background colors:

Reflexw uses some system colors for the background of the individual panels.

The following colors are used:

ActiveTitle, InactiveTitle, GradientActiveTitle, GradientInactiveTitle.

As these colors may be changed the visibility of some options might be restricted. In this case you should set the corresponding system colors to the default values shown below. Press the button combination [Windows] + [R], then select regedit and select ok. The registry menu will be opened. Select the options shown below and change the corresponding colors.



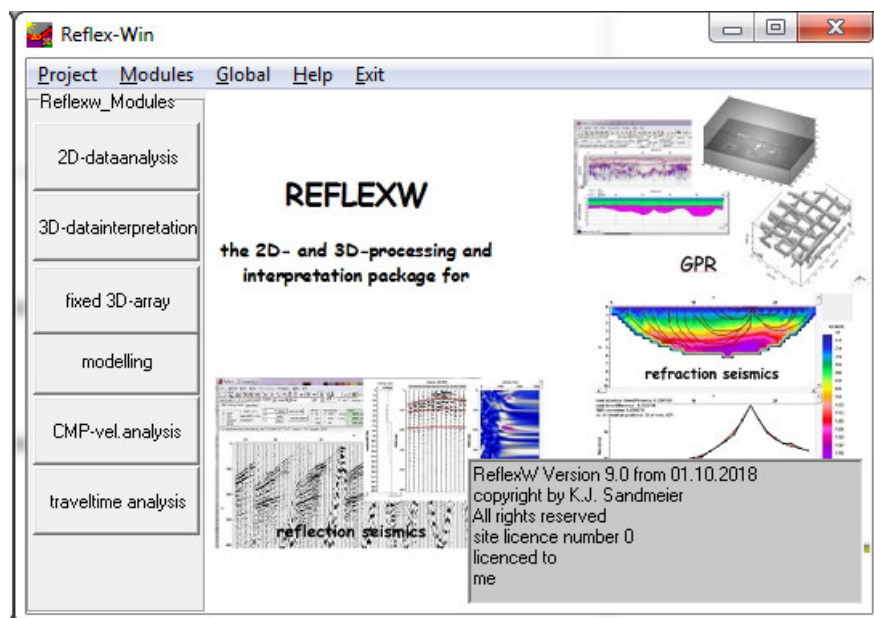
0. REFLEXW user's guide

Within this short user's guide some basic applications are described. It mainly concerns:

- the import and display of GPR-data (chap. 0.3)
- the filtering (chap. 0.4)
- the picking (chap. 0.5)
- the use of GPS-coordinates (chap. 0.6.2)
- the seismic refraction interpretation (chap. 0.7)
- the seismic reflection interpretation (chap. 0.8)
- the 3D-datainterpretation (chap. 0.9)
- the modelling/tomography (chap. 0.10).

0.0 the different REFLEXW modules

This is the first window of REFLEXW from which you may enter the different program modules (see also program modules).



Project: Selection of a new project. After selecting this option the directories included in the current project directory are listed. From these you can choose the project you want to work with.

Exit: Exit the program REFLEXW.

Modules: Chose of the different program modules:

- **2D data-analysis** which allows the complete processing of 2D-lines.
- module **CMP velocity analysis** which the calculation of a one-dimensional velocity-depth-distribution from CMP- or moveout-data based on different analysis techniques.
- module **3D data-interpretation** for displaying x-,y- or z-slices. The slices are either displayed in manually scalable windows or by moving through the 3D-cube using a track bar.
- module **fixed 3D-array** - allows the interactive interpretation of multichannel data with a fixed offset of the individual channels and based on relative coordinates. The target of this module is the interpretation of data consisting of many traces into one direction, e.g. along a road and a relatively small number of datapoints perpendicular to it (number of parallel lines).
- module **modelling** for the 2D-simulation of the seismic or electromagnetic wave-propagation based on a Finite Difference approximation. In addition a tomographic algorithm for travelttime data is included.
- module **travelttime analysis 2D** which allows to analyse and interpret picked first arrivals (refraction seismics).

If the main windows will be minimized it is possible to maximize them again using the corresponding button within the main menu.

0.1 Directory structure/project directory

The program REFLEXW uses a special directory structure, where the data are stored in the order of projects and processing steps.

The root project directory name is the subdirectory, where all projects shall be stored - e.g c:\data\reflex.

The program directory is the directory where the program has been installed.

It is not recommended to use the folder C:\Program Files (x86)\reflex as root project directory because of the virtualization of this directory. If used nevertheless the files can be found under C:\Users\username\AppData\Local\VirtualStore\Program Files (x86)\Reflex, where username is the name of the logged-in user. A warning message appears if the project directory has been chosen under the path C:\PROGRAM FILES (X86).

A new root project directory may be created manually, e.g. using the explorer or using the button **new root directory** within the ProjectDirectory menu. For that purpose please you first have to select the overall root (e.g C:\ if your root directory shall be saved directly under C:\) and then click on new root directory. Now you may create a new project directory by entering the project root directory name.

The last specified root project directory is automatically stored when leaving the program.

Under the root project directory the user has to create himself his projects as subdirectories (option **new project**) These subdirectories are the so called project directories. All subdirectories created by the user under the root project directory are listed after calling REFLEXW.

An existing subdirectory can be chosen by simply click on the wanted directory (either single or double click).

When calling the option **confirm project** the current chosen directory is used as the working project directory.

When calling the option **new project** the current entered project name is used as the working project directory.

Each project again has 6 subdirectories, which, if not already created by the user, are created by the program REFLEXW automatically after selecting the project.

Overview over the directory structure:

	ASCII
	ORIGINAL
	ROHDATA
project root directory - project directory -	PROC DATA
	LINEDATA
	MODEL

Externally formatted data (e.g. SEG Y or SEG2-data) should be stored in the directory ASCII (default import directory) or ORIGINAL (default import directory if the option import from Original is activated) but it is also possible to import the data from any other directory or drive (see also Import Menu). The imported data will be stored under the path ROHDATA, the processed data are stored under

PROCdata. The LINEDATA directory contains the pick and layershow file. In the MODEL directory the files which form the base for a simulation are stored.

Installing the DEMO-DATA:

The demo-data enclosed on the CD are automatically installed under the path REFLEXW has been installed. Example:

installation directory: C:\Program Files (x86)\reflex

data directory: C:\Program Files (x86)\reflex\demodata

subdirectory rohdata:

mala_original_data.dat - original Mala data converted to Reflexw format

subdirectory procdata:

gprline_.00t - a GPR-2D profile with a 300 MHZ antenna -

this file may be used within the 2-d-data-analysis module.

seismic1.01t - a seismic shot file consisting of 46 shots -

this file can be processed using the CMP-processing routines within the 2-d-data-analysis module.

gprcmp1_.00t - a GPR moveout file - this file can be interpreted within the CMP-velocity- analysis module

3ddata_.00t - a 3D-GPR datafile - this file may be used within the 3D-datainterpretation module.

R001695-r001697 - 3 different seismic refraction shot files

subdirectory model:

gprmod - a model file for use within the modelling tool

tomo - a model file for use the tomographic inversion

subdirectory ASCII:

tomo1.tom - - ASCII-traveltime data for performing a tomographic inversion based on the initial model tomo.mod

gprline_.dzt - a dzt GPR-2D profile with a 300 MHZ antenna

xp005000.dzt - xp030000.dzt - some parallel dzt GPR files with a profile increment of 50 cm

dat_0001.rd3 - a ramac GPR 2D-profile

ramac_file.rd3 - a ramac GPR 2D-profile

xp*.dzt - 6 parallel GPR-files acquired with a GSSI system

shot_forward.sg2 - a SEG2 seismic forward shot

shot_reverse.sg2 - a SEG2 seismic reverse shot

subdirectory LINEDATA:

refrdata.pck - seismic refraction traveltimes put together and assigned to layers

The program has automatically copied the following demo data files to the program directory for the program Reflex3dScan:

Scan3DXY.sg2:

SEG2-formatted file containing a 3D-GPR dataset consisting of:

88 scans in x-direction and 35 traces/scan

antenna polarization in y-direction (perpendicular to the profile direction)

Scan3DYX.sg2:

SEG2-formatted file containing a 3D-GPR dataset consisting of:

35 scans in y-direction and 88 traces/scan

antenna polarization in x-direction (perpendicular to the profile direction)

0.2 some general comments

1. import NON-reflex formatted data:

Externally formatted data (e.g. SEGY or SEG2-data) should be stored in the directory ASCII (default import directory) but it is also possible to import the data from any other directory or drive. Use the option file/import within the 2D-dataanalysis module for importing the data. The converted files are written to the subdirectory ROHDATA, any processed data and the timeslices are saved under the subdirectory PROCDATA.

2. first filter operations:

Some data like MALA RD3, RD7 or PULSE-EKKO formatted data must be preprocessed in a special way in the module 2D data- analysis. The first filter is either the so called subtract-mean(dewow) under the 1D-filter processing group or the subtract DC-shift filter which eliminates a possible time constant shift. The second filter consists of the application of a gainfunction (e.g. option manual gain(y), energy decay, AGC or gain function under the Gain processing group). The AGC (AutomaticGain Control) or the energy decay has not to be used as a filter function but can also be used only for the presentation of the data (see plot options). Each filtered data set is stored on disc with a user defined label in the extension.

Using the option "change second to primary" you may declare the current filtered file as the primary file which can be processed in a different manner. In order to enable a good control of the filtering you may choose between non split, vertically split or horizontally split window (options horizontal split or vertical split under plotoptions).

3. demodata

If the program is delivered on CD demo data are automatically installed under your working directory. Thereby a project named demodata is automatically created.

4 hardware dongle

if the program is supplied with a hardware dongle the dongle must be installed on one of the two parallel or on the USB port.

The program works both with the attached USB and the parallel port dongle. For a proper working of the hardware dongle under Windows9x or Windows NT and Windows 2000/XP the Crypto-Box Windows 95/2000/NT/XP driver must be installed.

Enter cbsetup.exe from the CD for the installation. Click on install and OK and then choose CRYPTO-BOX USB CryptToken (USB) for the USB dongle or CRYPTO-BOX Versa (LPT) for the parallel port dongle.

The files CBNDDL.DLL and CBN.VXD are copied under your system directory.

If the installation is made under Windows NT or Windows 2000/XP, in addition three different drivers (MARXDEV1.SYS, MARXDEV2.SYS and MARXDEV3.SYS) are installed which allow the proper access to any parallel port.

To be considered under WindowsNT and Windows 2000/XP: You must be a member of the administration group for installing the driver properly.

5. display resolution

Please use high color (16 bit) resolution because this is the best compromise between color resolution and needed RAM. True color (32 bit) needs twice as much memory and therefore slows plotting significantly. When using 256 colors the colors don't remain when switching to another window.

6. Decimal separator

Reflexw always uses the dot (".") for the numerical data representation within ASCII-files to be read or saved. The decimal separator within the Reflexw input windows is given by default by the Windows settings and may be changed within the global settings menu (chap. 1.2.1).

7. data restriction

max. number of processing steps: 40

modelling - interactive construction of layer boundaries:

max. number of layers: 500

max. number of layer points per layer: 250

32 bit version:

max. number of traces: 1048576

max. number of picks: 1048576

max. number of samples: 65000

max. number of traces for fk-filter: 32500

3D-datainterpretation:

max. number of datapoints for 3D-files in each direction: 2048

64 bit version:

max. number of traces: 3000000

max. number of picks: 1048576

max. number of samples: 65500

max. number of traces for fk-filter: 325000

3D-datainterpretation:

max. number of datapoints for 3D-files in each direction: 8192

8. Licence number and password

The licence number and the password will be stored within the file password.txt. This file will not be removed when uninstalling Reflexw. If for some reasons a new Reflexw version with a different licence number/password will be installed on the same path the new access parameters will not be queried. You have the possibility under Help/change lic.nr./password within the Reflexw main menu to change the licence number and password and thereby to enter the correct ones.

0.3 Import and first display of GPR-data

In the following the import and the first display of GPR-data within REFLEXW are described. The functions are described using a MALA/RAMAC GPR-example file. You only have to change some options for other data types.

0.3.1 Import GPR-data

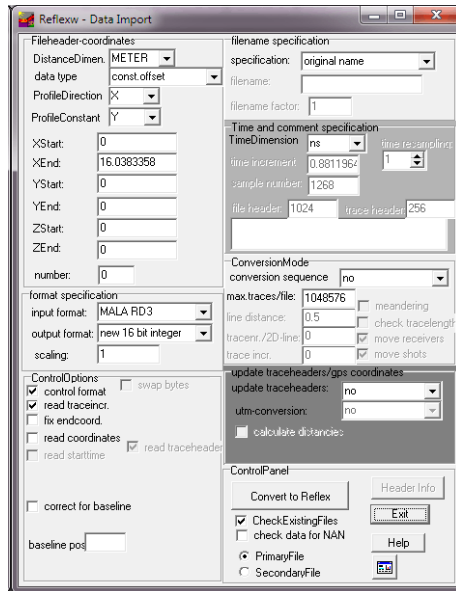
1. enter the module **2D-dataanalysis**

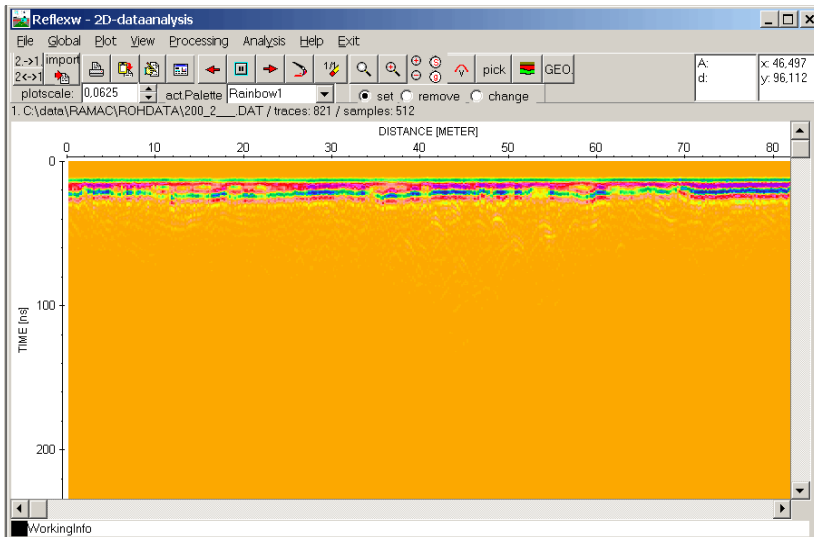
2. enter the **import** menu using the option File/Open/import. The REFLEXW-DataImport menu appears (see figure at the right).

3. Make the following inputs:
 input format: MALA RD3
 output format: 16 bit integer
 filename specification: original name for example; to be considered: REFLEXW (until version 2.5.x) only uses max. 8 (16 from version 3) characters for the filename for DOS-compatibility. Choose X or Y as ProfileDirection and Y or X as ProfileConstant. Choose if the traceincrement and/or the coordinates shall be read from the original data.

4. Activate the option **Convert to Reflex**. A fileopen menu appears with the directory ASCII under your project directory as the standard import path. You may choose an original MALA RD3 file (RD3 or RAD-file) from this import path or from any other directory. In any case both MALA RD3 files (RD3 and RAD) must be present. After having chosen the wanted MALA RD3 file the data are converted into the REFLEXW internal format and stored under the path ROHDATA under your project directory. With the option PrimaryFile activated the imported data are automatically displayed into the primary window (see figure below).

5. Exit the import menu using the option exit.





0.3.2 Display the data

1. After having done the import the data are displayed using the standard plot options. You may change these plot options using the option Plot/Options. Activating this option the Plot Options menu appears (see figure on the right).

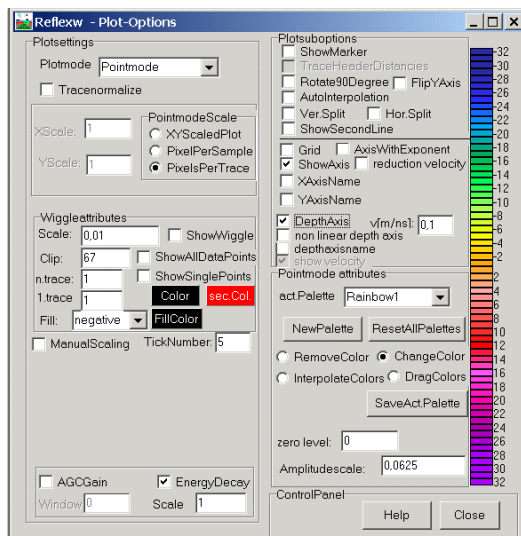
2. The main plot options for GPR-data are:

- Plotmode
- PointmodeScale
- EnergyDecay
- AmplitudeScale

Plotmode: by default use Pointmode for GPR-data.

PointmodeScale:

XYScaledPlot: the data are completely plotted into the current window provided that the two scale options XScale and YScale are set to 1. The option may be used for small data (few traces) or if you want to display the complete dataset into the main menu (the Zoom-options are available - see also option Xscale and Yscale)). With no zooming and large datasets the display



resolution is quite poor.

PixelPerSample: the plotting size of each data point is given in screen pixels. Zooming or rescaling is only possible in y-direction (option Yscale). The option might be useful for large data.

PixelsPerTrace: the distance between successive traces is given in screen pixels. The complete time series of each trace is plotted corresponding to the size of the current window. No zooming possibilities are available. For example this option may be used for large data (many traces).

EnergyDecay: the MALA RD3 GPR-data are raw data without any gain in time-direction. Therefore activate this option if no gain-filter (see also filtering guide) has been applied on the GPR-data in order to compensate the energy decay with time.

AmplitudeScale: this parameter controls the contrast of the data display.

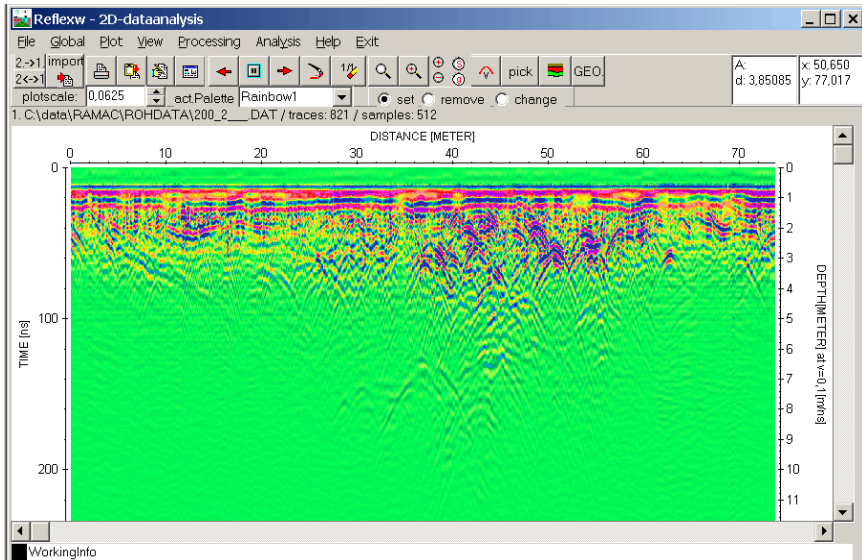
On the lower figure you see the data using the plot parameters shown in the plotoptions figure above.

To be considered: Normally the data must be filtered (see the filtering guide). The main filters are:

static correction to compensate for the time delay of the first arrival.

y(time)-gain

dewow



0.4 Filtering the imported data

In the following the filtering within REFLEXW is described including single filtering (chapter 0.2.3.2) and sequence processing (chapter 0.2.3.3).

0.4.1 General overview of filtering

1. The filtering options can only be applied on REFLEXW formatted data - this means that other formatted data (like MALA RD3, GSSI, SEGY,...) must be imported first using the option File/import. The imported data are stored under the subdirectory rohdata.
2. The filtered data are always stored on a separate file using the same filename and a special extension given by the processing label. Example: original file: profile1.dat; filtered file: profile1.00t (the processing label was set to 0). The filtered data are always stored under the subdirectory procddata.
3. Filtering can be done for a single dataprofile (see section II) or for a number of profiles (so called sequence processing, see section III).

0.4.2 Filtering of a single datafile

In the following the filtering of a single REFLEXW-formatted datafile is described step by step. The following filtering sequence is used:

subtract-mean(dewow)

static correction

manual gain(y)

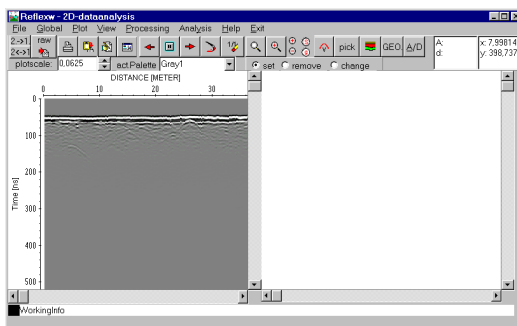
background removal

This may be a standard filter sequence for GPR-data which are stored as raw data like MALA RD3 or PULSEEKKO data. For a detailed information about the individual filter steps please press F1 within the corresponding filtermenu.

1. enter the module **2D-dataanalysis**

2. **load** the wanted datafile into the primary window using the option File/Open/rawdata and then choose the wanted datafile.

3. The chosen datafile has been plotted into the primary window (Hor.Split activated). Depending on the settings of the split plot options (see option Plot/Options - options Ver.Split



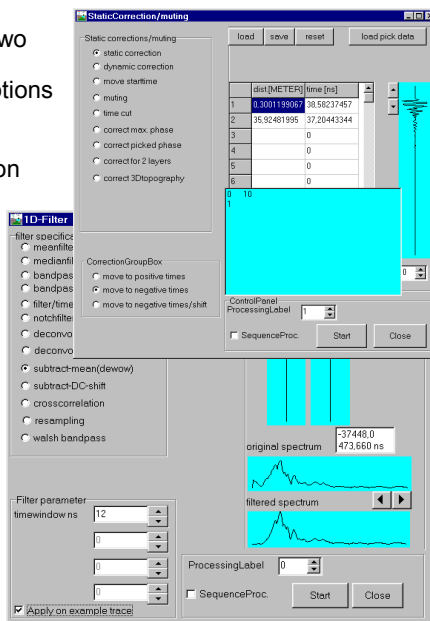
or Hor.Split activated) the screen is subdivided vertically or horizontally into two different windows (primary window and secondary window). If none of the two options is activated the screen is not subdivided.

4. enter the **first** filter step using the option **Processing/1D-Filter**.

5. The **1D-filter** window appears. Enter the following parameters or options:

- activate the option **subtract-mean (dewow)**
- enter the filterparameter **timewindow**
- enter the wanted **ProcessingLabel**
- **start** the filtering

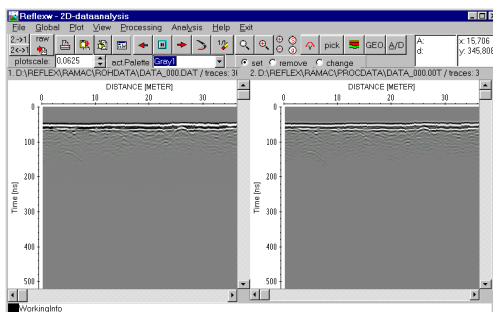
With the option **Apply on example trace** activated the effect of the entered filterparameter(s) can be controlled interactively (filtered trace and filtered spectrum). Trace number controls which trace is shown.



With the option **SequenceProc.**

activated the SequenceProcessing menu opens in addition which allows you to add the current filter step to the processing flow (see chapter 0.2.3.3).

6. The filtered data have been plotted into the secondary window. Now **close** the filter window and you must enter the option **File/ChangeSecondToPrimary** in order to use the filtered dataset for the next desired filter step.



7. Enter the **second** filter step using the option **Processing/staticCorrection/muting**.

8. Within the **StaticCorrection/muting** window activate the option **static correction** and the suboption **move to negative times**.

9. A table appears on the right side. You may either enter manually the correction values within this table or you may enter the values interactively using the left mouse button within the data. The option save allows to store the correction values on file for any later step.

10. Enter the wanted **ProcessingLabel** and **start** the static correction. To be

considered: the processing label must be different from that of the primary file because original and filtered datafile must not have the same filename.

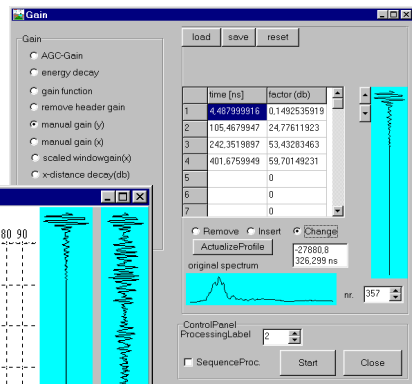
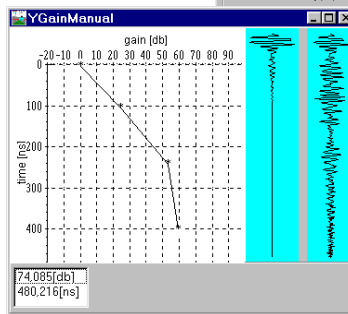
11. The static corrected data have been plotted into the secondary window. **Close** the static correction window and use the option **File/ChangeSecondToPrimary** in order to use the filtered dataset for the next desired filter step.

12. Enter the **third** filter step using the option **Processing/Gain**.

13. Within the Gain window activate the option **manual gain (y)**. A table appears on the right side for entering the gainvalues (analogous to the previous static correction) together with a new window which allows to interactively enter the gainvalues.

14. Enter the wanted **ProcessingLabel** and **start** the manual gain (y) processing step. Again the processing label must be different from that of the primary file.

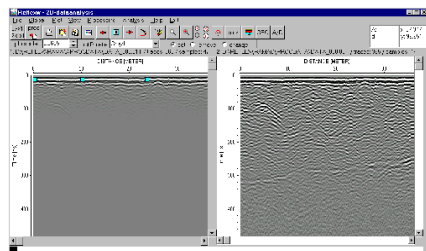
15. The manual gain (y) corrected data have been plotted into the secondary window. **Close** the gain window and enter the option **File/ChangeSecondToPri** the filtered dataset for step.



mary in order to use the next desired filter

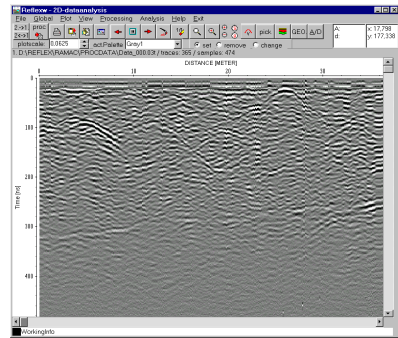
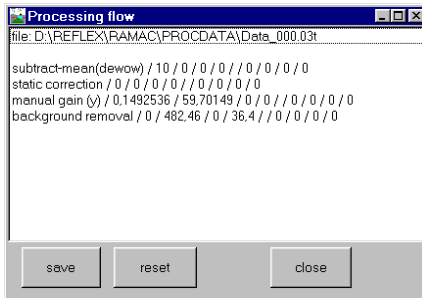
16. Enter the **last** filter step using the option **Processing/2D-filter**.

17. Within the **2D-filter** window activate the option **background removal**. In this case the complete datarange is used for determining the filtertrace (suboption whole line activated). Enter the wanted **ProcessingLael** and **start** the background removal filter. Again the processing label must be different from that of the primary file.



18. The filtering is finished. You may use the option

File/ChangeSecondToPrimary and deactivate the option **Hor.Split** within the **plotoption** menu in order to only view the finally filtered dataset. The complete processing flow can be displayed using the option **File/ProcessingFlow**. The processing flow of the current file may also be saved on a separate file and be used for a further sequence processing (see section 0.2.3.3).



0.4.3 Filtering different 2D-lines using the same filter sequence (sequence processing)

The sequence (batch) processing facilitates a completely automatic sequence of processing steps for a choosable number of profiles. For this you have to define a sequence of processing steps before, then select the data lines and then start the sequence processing.

1. Defining the sequence during the filtering of a single example file (see section II).

For that purpose you must always activate the option **SequenceProc.** when entering any of the filter menus. Then the **SequenceProcessing** menu opens in addition and you are able to add the current filter step to the processing flow using the option **AddCurrentProc.**

The step AddCurrentProc. must be done after having specified the filter parameters and before closing the current filter menu (e.g. between step 5 and step 6 within chapter II).

After having done all wanted filter steps for the example file including adding to the sequence processing you may save the current sequence processing on a separate file using the option

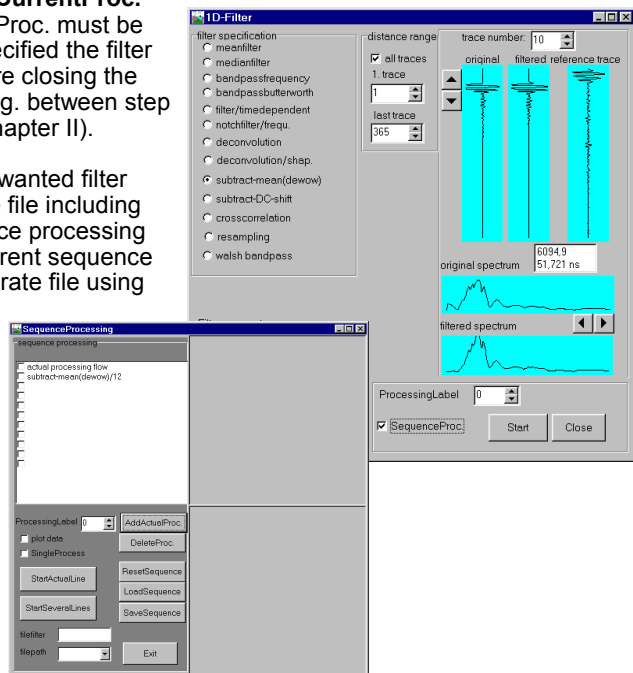
SaveSequence.

Such a processing sequence file may be loaded at any time using the option **LoadSequence.**

Normally the current processing step is added at the end of the processing sequence but it is also possible to include it at any

position. For that purpose you must click on the wanted processing step (a tick appears) the current step shall be included before.

It is also possible to remove a special processing step from the processing flow. For that purpose you must click on the wanted processing step (a tick appears) and then use the option **DeleteProc.** in order to remove this processing step. To be considered: for some processing steps where you have to enter the processing values by the mouse or within the table these values have to be saved on an ASCII file before defining/starting the sequence processing. For the



following processing steps the edit values are read from a file:
manual gain (y), manual gain (x), static correction, muting and fk-filter.

Therefore when you are adding for example the processing step static correction to the sequence processing flow first you must enter the wanted static correction values and then you must press the option save within the static correction menu. The option save stores the current edit values on an ASCII file under the project directory with the name correct and a freely choosable extension. This datafile is asked for when you are starting the sequence processing which includes the static correction (see item 2).

2. Start the sequence processing

The sequence processing menu is entered either during filtering a single example file or by the option **Processing/SequenceProcessing**.

After having defined the processing flow you must specify the **ProcessingLabel** and then start the sequence processing.

Activating the option **StartCurrentLine** applies the processing flow on the current primary file.

Activating the option **StartSeveralLines** applies the processing flow on a number of files to be chosen. The option **filefilter** allows to enter a filter for the multi file choice. **Filepath** specifies the path for loading the files to be processed. The multifile choice is done using the Shift or ctrl-key.

Activating the option **plotdata** allows you to display original and filtered file simultaneously, whereby a direct control of the result is given.

You may choose between the so-called sequence mode and the single processing mode (option **SingleProcess** activated) with the possibility of applying the processing steps individually on the original file(s).

Flow for the option **StartSeveralLines**:

1. After having defined the sequence processing (see item 1.) activate the option **filepath** and choose **rohdata** if the files to be processed are the original data or **procddata** if the files have been processed before.
2. Enter the wanted processing label.
3. activate the option **plot data** in order to plot the processed files during the sequence processing.
3. Click on **StartSeveralLines** and choose the wanted files - multifile choice using the shift or the ctr-key.
4. the sequence processing is started and after having finished the processing a report menu is shown.

To be considered: For the following processing steps the edit values are read from an ASCII file which is asked for after having choosen all wanted files (item 4):

manual gain (y), manual gain (x), static correction, muting and fk-filter.

The program uses the correction (processing) values for each profile within the sequence processing. Therefore it must be ensured that the precondition for that holds true (especially for the processing step static correction).

0.4.4 Handling of a strong topography for ZeroOffset data

If a strong topography is present the following peculiarities must be considered:

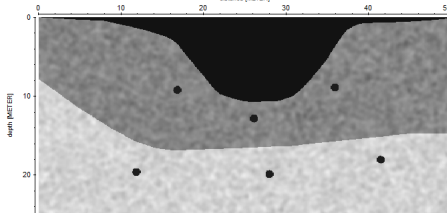
- the two-way traveltimes section does not represent the true location. For example the vertices of the diffractions mounted below the flanks of the topographic surface will be clearly dislocated as the back-reflected beam, which is the shortest duration, does not point in a purely vertical direction but perpendicular to the topographic surface. This shift is greater, the deeper the object is located.
- the distance axis if measured along the topographic surface has been lengthened and does not represent the correct x-projection.

Intensive studies using FiniteDifference simulations for different models (see our article www.sandmeier-geo.de/Update/simulations/Strong_topography.pdf) have shown that the topographic migration in combination with the option correct 3D-topography yields the best results in the presence of a strong topography. Therefore the following guide only treats these two processing options.

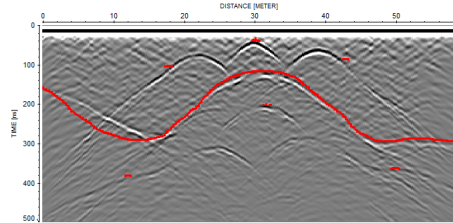
As both the topographic corrections values and the ZO data can be acquired in a different way in the following the sequence processing for a best correction of the topography for different measure conditions are given.

1. The data have been equally spaced acquired along the topography and the topographic values have been measured as xyz-values

1. F:\data\migration_topo\PROC\DATA\MULDE_FLUCT.001 / traces: 948 / samples: 565



2. F:\data\migration_topo\PROC\DATA\M_F_00001EY.001 / traces: 292 / samples: 416

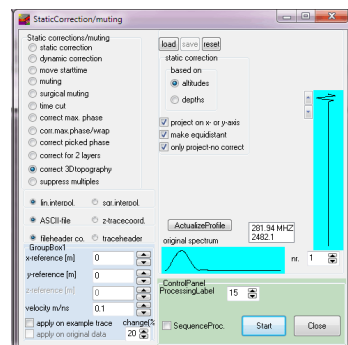


The left panel shows the model of the underground. The source/receiver has been equally spaced along the topographic interface. As a result (right panel) the distance axis has been lengthened (from 50m to about 58 m). In addition the vertices of the diffractions which are placed below the flanks of the topographic surface are clearly dislocated.

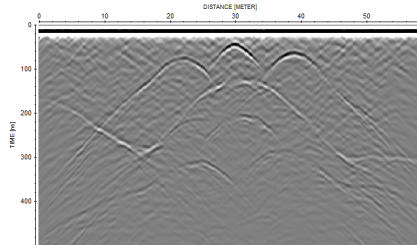
In contrast to the GPR data the topographic values have been measured as xyz-values with x corresponds to the true profile position in x-direction, y is equal 0 and z represents the elevation relative to 0.

0.000000	0	0.000000	29.212790	0	-10.632910
4.674047	0	-0.303798	30.996309	0	-9.949367
7.872079	0	-0.455696	32.410820	0	-8.582278
12.730630	0	-1.594937	34.747849	0	-5.696202
15.498160	0	-2.506329	36.223862	0	-3.265823
16.871920	0	-3.458647	37.330879	0	-1.670886
17.650681	0	-4.556962	38.238918	0	-1.127820
19.618700	0	-6.987341	38.854679	0	-0.902256
21.059111	0	-8.872180	39.544899	0	-0.759494
21.955721	0	-9.797468	44.403450	0	-0.607595
23.706900	0	-10.375940	47.601479	0	-0.379747
24.815269	0	-10.676690	49.754002	0	-0.151899
25.830259	0	-10.784810	50.000000	0	-0.151899

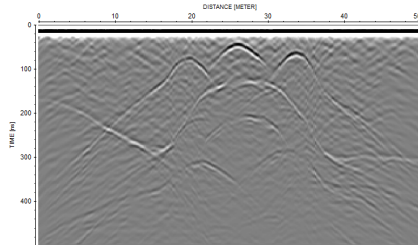
As the topographic migration needs an identical level both for the coordinates of the GPR data and of the topographic correction values the first processing step must be the projection of the distance axis measured along the topographic surface. This is done using the option **correct 3D-topography** with activated suboption **only project-no correct** based on the topographic correction ASCII-file (see above). As a result (see next picture) the distance axis has been shortened and the diffractions below the flanks are clearly deformed.



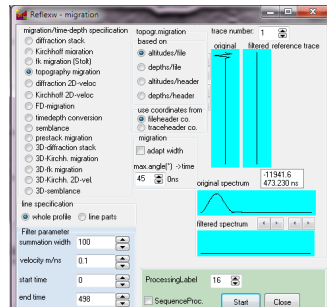
1. F:\DATA\MIGRATION_TOPO\PROC\DATA\M_F_00001EY.031 / traces: 252 / samples: 416



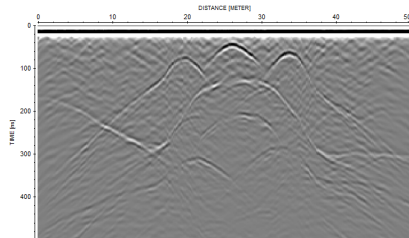
2. F:\DATA\MIGRATION_TOPO\PROC\DATA\M_F_00001EY.151 / traces: 252 / samples: 416



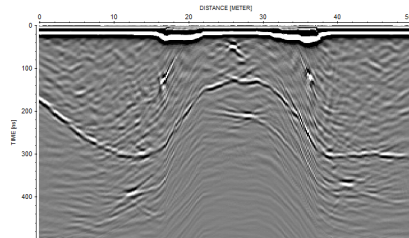
The next step is the **topographic migration** based on a constant velocity and a selectable summation width. Again the topographic correction ASCII file is used. As a result (see below) the diffractions are well focused and the reflector has been moved to its correct place but still following the topographic surface.



1. F:\DATA\MIGRATION_TOPO\PROC\DATA\M_F_00001EY.151 / traces: 252 / samples: 416

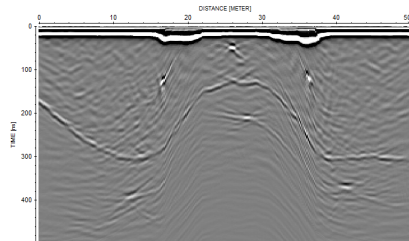


2. F:\DATA\MIGRATION_TOPO\PROC\DATA\M_F_00001EY.161 / traces: 252 / samples: 416

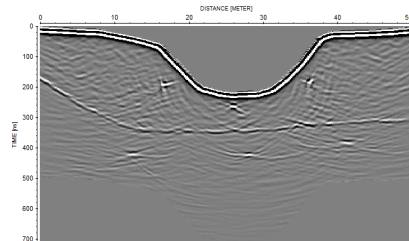


The last step is the topographic correction using again the option **correct 3D-topography** but with deactivated suboptions. The result is very similar to the input model.

1. F:\DATA\MIGRATION_TOPO\PROC\DATA\M_F_00001EY.161 / traces: 252 / samples: 416

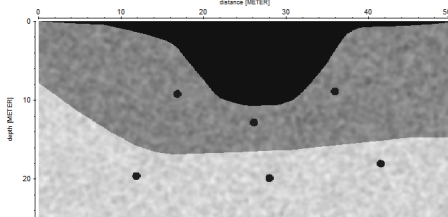


2. F:\DATA\MIGRATION_TOPO\PROC\DATA\M_F_00001EY.171 / traces: 252 / samples: 596

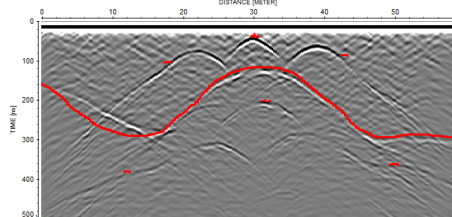


2. The data have been equally spaced acquired along the topography and the topographic values have also been measured along the topography

1. F:\data\migration_topo\PROC\DATA\MULDE_FLUCT_001 / traces: 948 / samples: 565



2. F:\data\migration_topo\PROC\DATA\M_F_00001EY_001 / traces: 292 / samples: 416

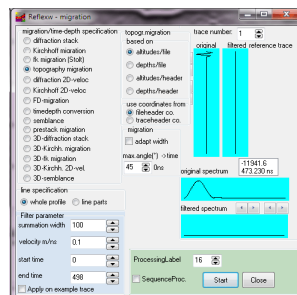


The left panel shows the model of the underground. The source/receiver has been equally spaced along the topographic interface. As a result (right panel) the distance axis has been lengthened (from 50m to about 58 m). In addition the vertices of the diffractions which are placed below the flanks of the topographic surface are clearly dislocated.

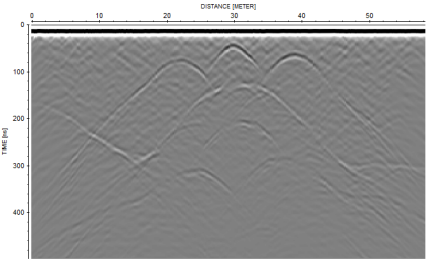
The topographic values have been measured as xyz-values with x corresponds to the profile position along the topography, y is equal 0 and z represents the elevation relative to 0.

0.000	0.0	0.000000	30.000	0.0	-10.752213
2.000	0.0	-0.132743	32.000	0.0	-10.663716
4.000	0.0	-0.265487	34.000	0.0	-10.265487
8.000	0.0	-0.486726	36.000	0.0	-9.159292
10.000	0.0	-0.929204	38.000	0.0	-7.699115
12.000	0.0	-1.371681	40.000	0.0	-6.150443
14.000	0.0	-1.946903	42.000	0.0	-4.469027
16.000	0.0	-2.610620	44.000	0.0	-2.787611
18.000	0.0	-3.849557	46.000	0.0	-1.327434
20.000	0.0	-5.442478	48.000	0.0	-0.752212
22.000	0.0	-6.991150	50.000	0.0	-0.707965
24.000	0.0	-8.584071	54.000	0.0	-0.530973
26.000	0.0	-9.911505	56.000	0.0	-0.353982
28.000	0.0	-10.530973	58.000	0.0	-0.132743

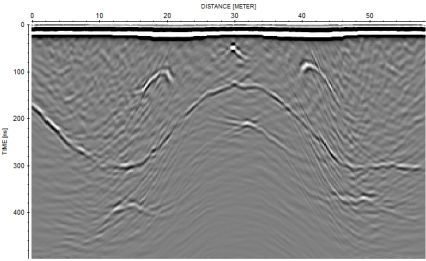
As the base for the coordinates are the same both for the GPR data and the topographic correction values one can directly perform the **topographic migration** based on a constant velocity and a selectable summation width. The ASCII topographic correction file with the x corresponding to the profile position along the topography is used. As a result (see below) the diffractions are quite well focused and the reflector has been moved to its correct place.



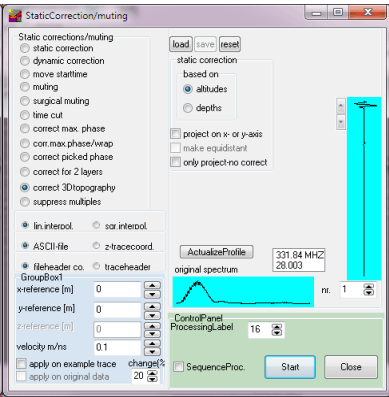
1. F:\DATA\MIGRATION_TOPO\PROC\DATA\M_F_D0001EY.031 / traces: 252 / samples: 416



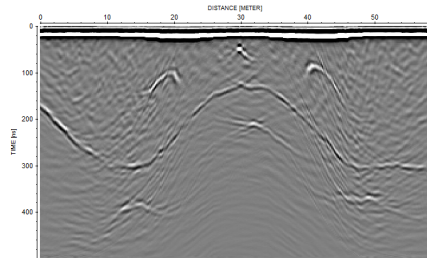
2. F:\DATA\MIGRATION_TOPO\PROC\DATA\M_F_D0001EY.151 / traces: 252 / samples: 416



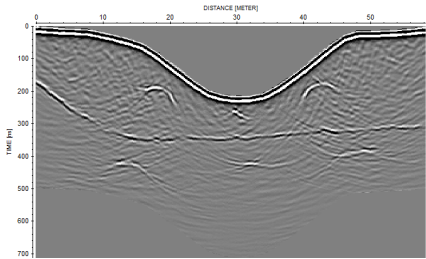
The last step is the topographic correction using the option **correct 3D-topography** with deactivated suboptions based on the same ASCII topographic correction file. The distance axis of the resulting profile still represents the positions along the topographic interface. The diffractions below the flanks are more poorly resolved but located at the correct place. The layer and the diffractions below the trough are well resolved.



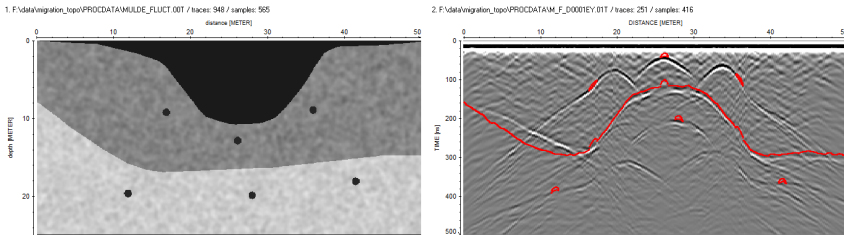
1. F:\data\migration_topo\PROC\DATA\M_F_D0001EY.151 / traces: 252 / samples: 416



2. F:\DATA\MIGRATION_TOPO\PROC\DATA\M_F_D0001EY.161 / traces: 252 / samples: 595



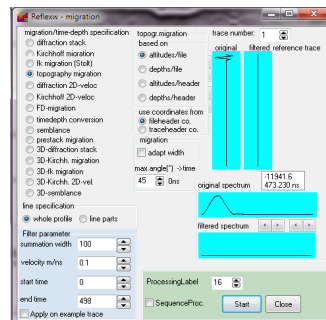
3. The data have been equally spaced acquired along the true x-axis and the topographic values have been measured as xyz-values



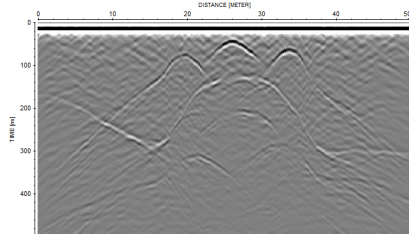
The left panel shows the model of the underground. The source/receiver has been equally spaced along the true x-axis. The distance axis of the model and of the GPR data are the same. The vertices of the diffractions which are placed below the flanks of the topographic surface are clearly dislocated. The topographic values have also been measured as xyz-values with x corresponds to the true profile position in x-direction, y is equal 0 and z represents the elevation relative to 0.

0.000000	0	0.000000	29.212790	0	-10.632910
4.674047	0	-0.303798	30.996309	0	-9.949367
7.872079	0	-0.455696	32.410820	0	-8.582278
12.730630	0	-1.594937	34.747849	0	-5.696202
15.498160	0	-2.506329	36.223862	0	-3.265823
16.871920	0	-3.458647	37.330879	0	-1.670886
17.650681	0	-4.556962	38.238918	0	-1.127820
19.618700	0	-6.987341	38.854679	0	-0.902256
21.059111	0	-8.872180	39.544899	0	-0.759494
21.955721	0	-9.797468	44.403450	0	-0.607595
23.706900	0	-10.375940	47.601479	0	-0.379747
24.815269	0	-10.676690	49.754002	0	-0.151899
25.830259	0	-10.784810	50.000000	0	-0.151899

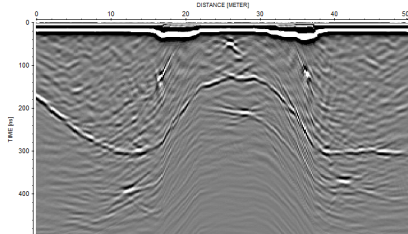
As the base for the coordinates are the same both for the GPR data and the topographic correction values one can directly perform the **topographic migration** based on a constant velocity and a selectable summation width. The ASCII topographic correction file with the x corresponding to the true x-position is used. As a result (see below) the diffractions are quite well focused and the reflector has been moved to its correct place.



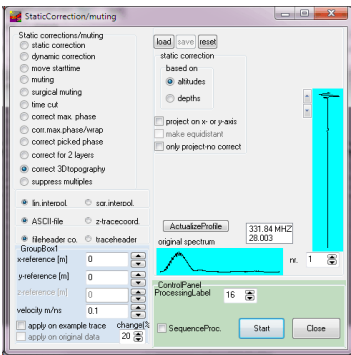
1. F:\DATA\IMGRATION_TOPO\PROC\DATA\M_F_D000EY157 / traces: 292 / samples: 416



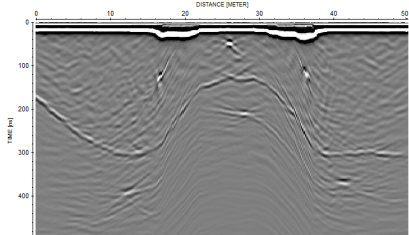
2. F:\DATA\IMGRATION_TOPO\PROC\DATA\M_F_D000EY157 / traces: 292 / samples: 416



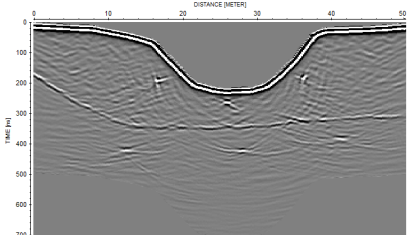
The last step is the topographic correction using the option **correct 3D-topography** with deactivated suboptions based on the same ASCII topographic correction file. The result is very similar to the input model. Both the diffractions and the layer are very well resolved.



1. F:\DATA\IMGRATION_TOPO\PROC\DATA\M_F_D000EY157 / traces: 292 / samples: 416



2. F:\DATA\IMGRATION_TOPO\PROC\DATA\M_F_D000EY171 / traces: 292 / samples: 596



0.5 Picking within REFLEXW

0.5.1 Picking single objects

0.5.1.1 Picking hyperbola along one profile line

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

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

0.5.1.2 Use of xy-coordinates defined within the fileheader

The ASCII-output described above does not contain the real xy-coordinates but the distance coordinates along the profile together with the start coordinate in the profile constant direction. Therefore these coordinates only correspond to the real xy-coordinates if the profiledirection is strictly orientated into x- or y-direction. If this does not hold true (e.g. diagonal lines) you must take into account the so called traceheader coordinates in addition.

The following steps are necessary before picking:

1. Define the geometry of your file within the **fileheader** either during the **import** or within the edit fileheader menu (option **edit fileheader**). The example shows a diagonal line.

2. After having defined the fileheader coordinates (do not forget so save them before when you have changed the fileheader coordinates within the fileheader menu) activate the option **ShowTraceHeader**.

3. The traceheader menu opens. Here you may update the traceheadercoordinates either based on the fileheader coordinates or based on different ASCII-files (see also guide for GPS-coordinates, chap. 0.6). Enter within the Update panel for **type** fileheader and press the button **update**.

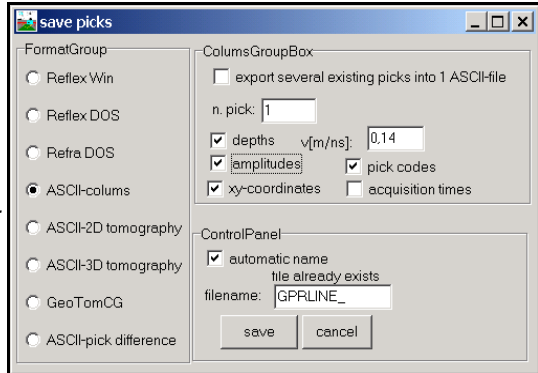
4. Now the traceheader coordinates are defined based on the given fileheader coordinates. You may check the coordinates using the **trace number** edit button.

5. Close the traceheader and the fileheader menu and pick the hyperbola like described in chapter 1.

6. When saving the picks using the format ASCII-columns you must activate the option **xy-coordinates** in addition.

7. 12. Now each line of the output file contains the following informations:
*profiledistance profileconstant-
 start shot-x shot-y receiver-x
 receiver-y traveltime depth
 amplitude code*

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



0.5.1.3 control and output of single picks for several profiles

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

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

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

3. Choose the wanted filepath and press show all lines

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

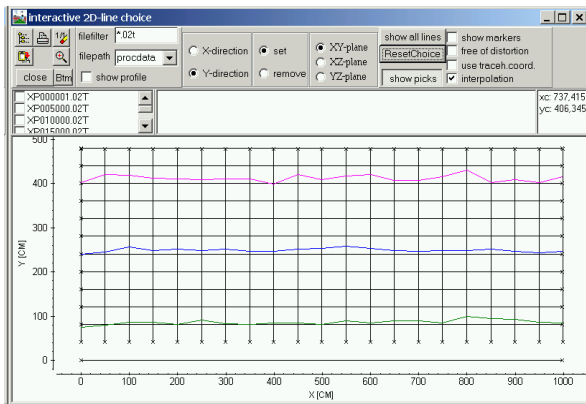
5. Picks belonging to the same code are connected if the option interpolation is activated. If the option use code within the 2D-dataanalysis is activated the layershow-colors are used for the display of the picks.

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

7. Within the save pick menu you must choose the "ASCII-columns" pick format and you have to activate the option "export several existing picks into 1 ASCII-file".

8. Click on save and choose all wanted ReflexWin formatted pickfiles (multifilechoice using the shift or ctrl key).

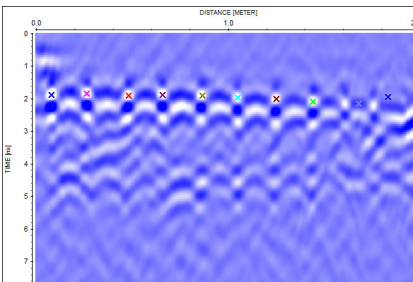
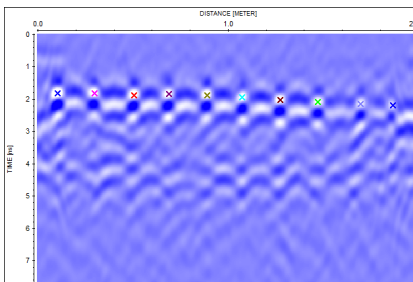
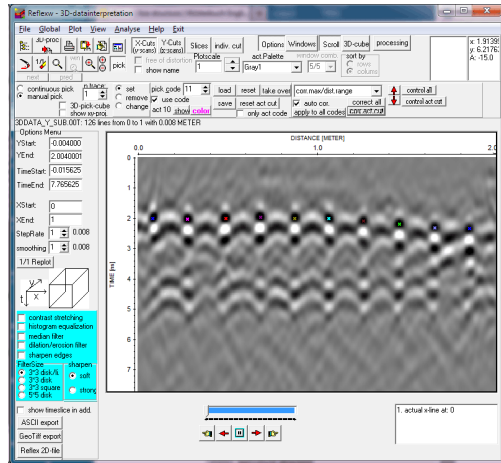
9. Each line of the resulting ASCII-file contains the following values:
traveltime depth amplitude x-shot y-shot z-shot x-receiver y-receiver z-receiver code(optional)
 Depths and amplitudes are set to zero if the corresponding options depth and amplitudes are deactivated.



0.5.1.4 Picking linear structures within a 3D-datafile

Reflexw allows a fast picking of single objects within the 3D-datainterpretation. The following procedure may be applied to all types of single objects like pipes, rebars or other line structures.

1. Load a 3D-file within the 3D-datainterpretation and choose Scroll and the appropriate cut
2. Activate pick and activate use code and auto cor. based on corr.max/dist.range (option only act. code is deactivated)
3. Click on apply to all codes
4. Pick all elements using different codes (the F4 key may be used in order to increase the code) within the actual cut
5. Go to the next cut using the right scroll button (hand)
6. Click on take over - the picks from the previous cut will be taken over to the actual cut
7. Click on the right arrow in order to scroll through the



complete 3D-block. The new picks will be displayed. You can stop the scrolling at any time.

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

- enter the code of the false pick (here 9)
- activate the option only act code
- change the correction type if necessary
- activate change and click on the correct pick position

- activate the option take over and scroll using the right arrow to the end of the block, the wrong picks will be changed according to the new settings.

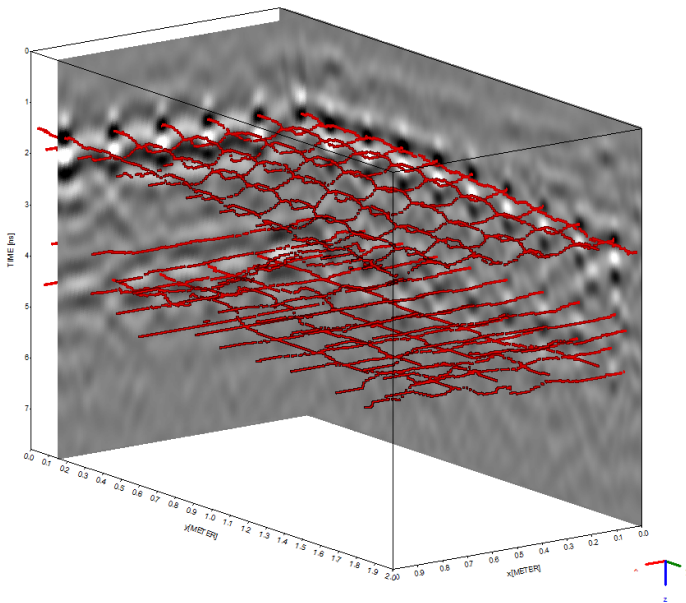
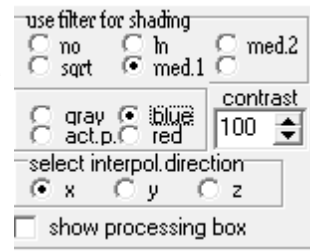
9. Make the same for all wrong picks
10. Save the picks
11. Deactivate the option only act code and reset all picks using the option reset
12. Enter a new code for the starting pick of the next set of picks, e.g. the lower picks. Make sure that the new pick code(s) has not been used before.

13. Perform step 4 to 10 to the next set of picks.
14. Perform step 11 to 13 for all wanted pick sets.

15. It is recommended to store all different picks sets on different files as this simplifies the semi automatic picking.

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

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



0.5.2 Picking different reflectors within REFLEXW

0.5.2.1 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. 0.5.2.2)
2. save the picks of one reflector using a special name (e.g. layer 1)- enter the layer number, the mean velocity and a layer code
3. repeat step 1 and 2 until all reflectors have been picked
4. enter the layer-show
5. create a velocity file if layer velocities shall be used instead of mean velocities (chap. 0.5.2.3)
6. create the layer-show (chap. 0.5.2.4)
7. create a report (chap. 0.5.2.5)

0.5.2.2 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)

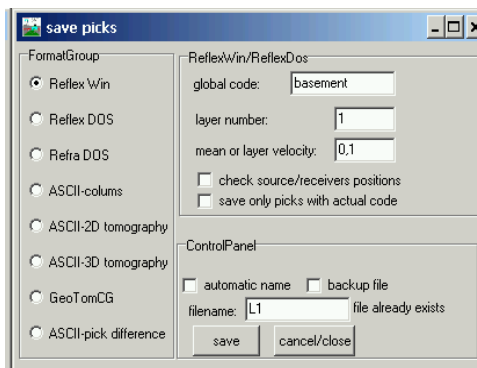
5. use the option **save** for saving the picks of one reflector using the format Reflex Win. Enter the following parameters:

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.

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

For the **time-depth conversion** using the option create LayerShow (see section 0.5.2.4) it is necessary to take a **velocity** into account.

The first possibility is to use the **constant mean velocities** for the total overburden or **layer velocities** of every pick file, which were set during the pick process (see section 0.5.2.2 - step 5). If you want to use this option you may skip this chapter and you may continue with chapter 0.5.2.4.

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:

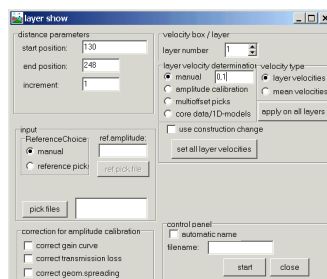
distance v-layer1 v-layer2 v-layer10 or v-layer 100 (from ver. 2.5)

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

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

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

2. there are 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.

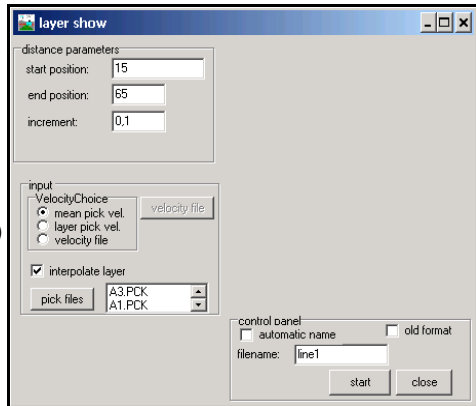
5. activate the option **close** for closing the create velocity menu.

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

0.5.2.4 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).

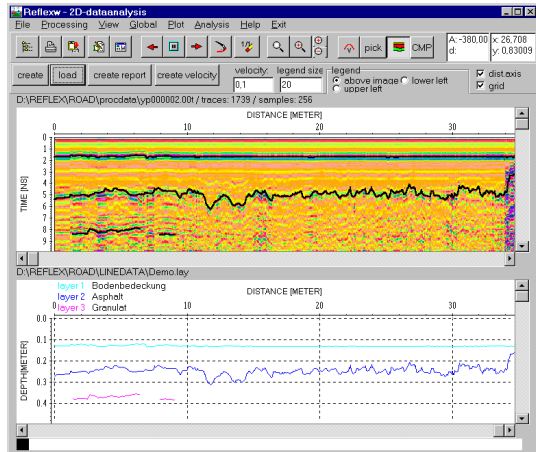
The picked traveltimes of the reflectors are transformed into depths using either a mean velocity for a distinct layer or based on a laterally varying velocity file.



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



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

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

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

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

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

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

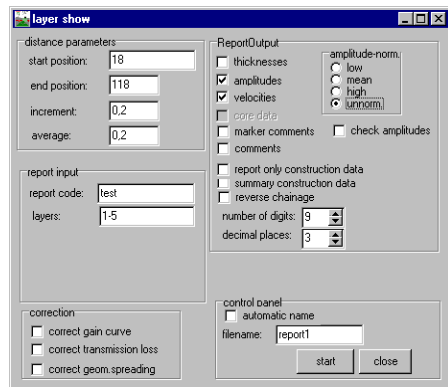
The results of the current layershow can be stored in a report. The output is performed either on a printer or on a file with an arbitrary name. According to the individual settings, the report contains information about single layers like e.g. thickness, velocity, amplitudes and coordinates. In addition,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

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

2. enter the necessary parameters for the output:

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

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



0.6 coordinates

Reflexw uses both fileheader and traceheader coordinates.

The fileheader coordinates contain all the information about the geometry of the whole file, e.g. it's start and end coordinates, distance dimension, trace increment. Therefore, if one deals with equidistant data, the coordinates of any single trace are known in relation to the other traces of one file without defining the traceheader coordinates.

The traceheader contains additional information concerning a single trace: e.g. shot position or receiver position in relation to the trace, which are needed in the case of multi shot data or if GPS-coordinates are present.

Normally the data are displayed using the fileheader coordinates. The traceheader coordinates are only used in addition, e.g. for the ASCII-output of the picks, for the CMP-processing of multi-offset data or for the interpretation of the refraction picked traveltimes.

0.6.1 peculiarities for meandering 2D-profiles

Reflexw only allows a positive traceincrement, this means the endcoordinates in profiledirection must be larger than the startcoordinate.

If you have acquired 2D-profile in two different directions (meandering acquisition) you have two different possibilities how to handle the problem:

1. during the import of the data:

- here you may enter a larger startcoordinate than the endcoordinate. In this case the imported profile will be automatically flipped in x-direction (this means the first trace becomes the last one and the last one the first one) and the end- and startcoordinate will be exchanged.

- if you have activated the option "parallel lines" you may activate the suboption "meandering". In this case each 2. line will be automatically flipped in x-direction. If your entered startcoordinate is smaller than the endcoordinate the 2., 4., profile will be flipped. If the startcoordinate is larger than the endcoordinate the 1., 3., 5., profile will be flipped.

The status menu shows after the import whether a profile has been flipped. Please check after the import whether the coordinates are correct and whether the correct profiles have been flipped because there are some additional controlling options like "read traceincr." and "fix endcoord." which affect the resulting coordinates. If the meandering option has been activated the traceheaders cannot be updated any more in most case because the original tracenumbers mostly form the base for the import of the gps data. In this case the profile must be flipped afterwards using the processing option XFlipProfile.

2. using the processing option "XFlipProfile":

this processing option can be found under processing/traceinterpolation /resorting. Use the option "XFlipProfile". The processed profile has been flipped in x-direction. The start- and endcoordinates will not be changed.

Within the sequence processing an option named only every 2.line exists which may simplify the file choice process.

0.6.2 Use of GPS-coordinates

REFLEXW allows to handle so called traceheader coordinates which are stored within the header of each trace. Therefore it is possible to use GPS-coordinates for a special analysis.

0.6.2.1 Set the GPS-coordinates

First you must set the **traceheader coordinates** using the **GPS-coordinates**. For that purpose it is necessary that the GPS-coordinates are stored within an ASCII-file. Several formats are supported. For example:

1. ASCII-DST format - each line contains the following informations:
tracenumber distance Shot-X-Pos Shot-Y receiver-X receiver-Y shot-Z(opt.) receiver-Z(opt.)

2. RAMAC GPS format (extension COR) - each line contains the following informations:

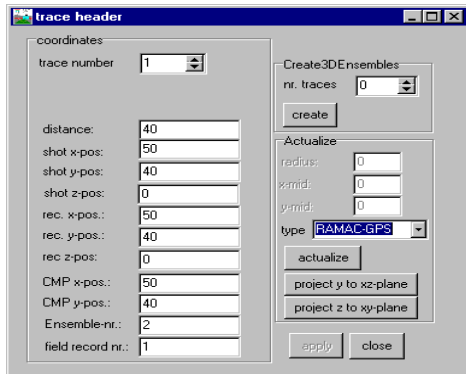
tracenumber,date,time,Northing,West,altitude,PDOP

The traceheader coordinates of REFLEXW are changed using one of these ASCII-formats within the **traceheader menu**. The traceheader menu is entered within the **fileheader menu** using the option **ShowTraceHeader**.

Within the **traceheader menu** you must choose either ASCII-file, ASCII-file/interpol. or RAMAC-GPS for type and then you must press the button update in order to update the traceheader coordinates. With the type "ASCII-file/interpol." activated an automatic interpolation is done if for various traces no coordinate data are present within the ASCII-file.

With the type "RAMAC-GPS" the program controls if there are consecutive identical coordinates in the *.cor file - these coordinates are automatically be replaced using an interpolation.

The option "interpolation 0-data" allows the interpolation of those coordinates which have 0 values.



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

To be considered: The traceheader coordinates are stored either using a 32 bit floating format (Reflexw formats 16 bit integer and 32 bit floating point within the import menu) or using a 64 bit double precision format (Reflexw formats new 16 bit integer and new 32 bit floating point) - see also chap. 1.5.4 Import format

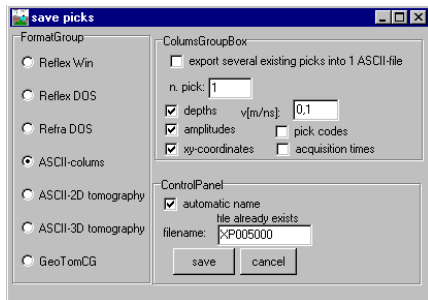
specification. If the coordinates have very high values with small changes, the data representation of the 32 bit floating format may not be good enough. In this case the Reflexw new format with the 64 bit double precision format must be used when importing the original data or a constant offset should be subtracted from the coordinates before storing them into the traceheaders.

0.6.2.1 Handle the traceheader(GPS)-coordinates

0.6.2.1.1 picking

The traceheader coordinates can be used for **exporting picked data** to an ASCII-file. For that purpose you must activate the option **xy-coordinates** within the save pick menu (format: ASCII-columns). Each line of the ASCII-file contains the following values: coordinate in profiledirection(10:3), coordinate in profileconstant(10:3), x-traceheader-coordinate, y-traceheadercoordinate, travel time(12:6), depth(10:4), amplitude(10:2).

The ASCII-file has the extension PCK and will be stored in the path ASCII in the current projectpath.



0.6.2.1.2 viewing options within the 2D-dataanalysis

Within the 2D-dataanalysis module there exist two different **viewing options** of the **traceheader xy-coordinates**.

First using the option **profile line (trace header coord.)** the profile location based on the traceheader coordinates is shown in an additional window (any curvature of the line coordinates is displayed). When moving the mouse cursor within the data window the current xy-position of the mouse cursor is also shown. Second with the option **TraceHeader axis** activated the xy-receiver traceheadercoordinates are displayed along the distance axis in addition.

The **traceheader z-coordinates** are used for the plotoption **correct header elevations**. If activated the traces are shifted based on the receiver and the shot elevation values stored within the traceheader of each trace and the entered elevation level. The shift levels are calculated from the difference of the entered elevation level and the individual traceheader elevation values. Based on the current velocity the travelttime shift value is calculated from the sum of the shot and the receiver elevation differences. The option is only enabled if the option elevation is activated.

The plotoption **TraceHeaderDistancies** allows to plot the profile based on the individual distances stored in the single traceheaders and not based on the equal trace increment of the fileheader. The option is available both for the wiggle and point plotmode.

0.6.2.1.3 processing options

There exist several processing options which allow the use of the traceheader-coordinates. For example:

a. **correct 3D-topography** under processing/static correction: With the suboption z-tracecoord activated the static correction is based on the shot and the receiver elevations stored within the traceheader of each trace. The option is available from Reflexw version 3.5.

b. **make equidist. traces**: the option allows to interpolate non-equidistant data in such a way that the resulting data are equidistant. The precondition is that the true position of each trace is stored in the parameter distance within the individual trace header. If the distance is not defined you may do this within the traceheader menu using the option update and the type "calculate distances" (from version 3.5).

0.6.2.1.4 3D-datainterpretation

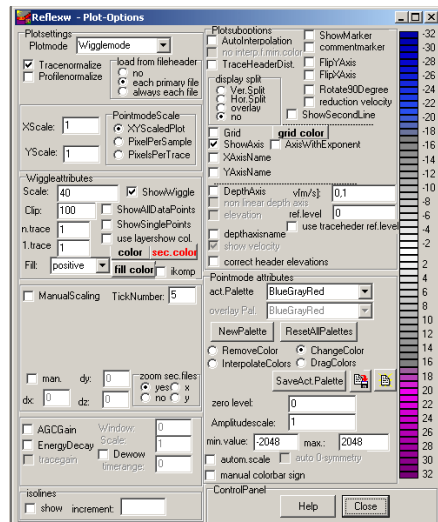
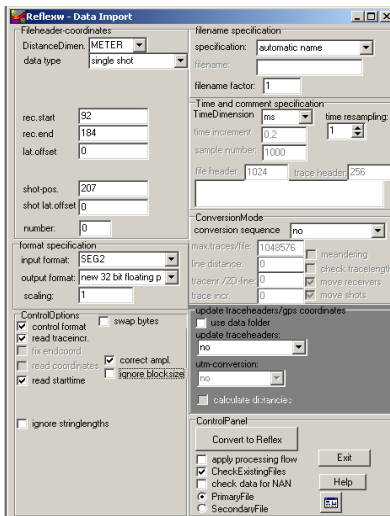
It is possible to build up a 3D-datafile based on freely distributed 2D-lines whereby the individual positions of the traces are either taken from the fileheader or from the traceheader coordinates (see also guide 3D-dataintepretation, chap. 0.9.1.2.2 - Generating a 3D-file for freely distributed 2D-lines). With the sorting type "receiver coordinates" the xy-positions of each trace is taken from the xy-receiver traceheader coordinates.

0.7 Introduction to seismic refraction data interpretation

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).

0.7.1 Import the data and pick the first onsets (done within the module 2D-dataanalysis)



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 options** (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.

edit trace header coordinates for F:\data\REFRDEMO\ROHDATA\R001695_DAT

trace-nr.	distance	shot-x	shot-y	shot-z	rec.-x	rec.-y	rec.-z	time delay	gain	time collect
1	92	160	0	0	92	0	0	0	1	0
2	96	160	0	0	96	0	0	0	1	0
3	100	160	0	0	100	0	0	0	1	0
4	104	160	0	0	104	0	0	0	1	0
5	108	160	0	0	108	0	0	0	1	0
6	112	160	0	0	112	0	0	0	1	0
7	116	160	0	0	116	0	0	0	1	0
8	120	160	0	0	120	0	0	0	1	0
9	124	160	0	0	124	0	0	0	1	0
10	128	160	0	0	128	0	0	0	1	0
11	132	160	0	0	132	0	0	0	1	0
12	136	160	0	0	136	0	0	0	1	0

TopographyGroupBox

topography (x,z values)

☒ update shot z-pos.

☒ update receiver z-pos.

☒ use x-traceheadercoord.

☐ apply x-z topography

☐ get distance along topography

EditGroupBox

apply borehole deviations

3D-view of boreholes

project on x

x <-> y y <-> z

source <-> rec.

rec. -> source

correct lat offset

smooth rec. z-coord.

smooth rec. xy-coord.

☐ smooth shot coord.

factor f. smooth: 4

interpolate

☐ non equidistant spread

☐ interpolate all

☐ update only shot coord.

fileheader coordinates

data type: single shot

shot-pos. 160

shot-offset 0

rec.start 92

rec.end 184

rec.offset 0

coordinate transformation

UpdateGroupBox

update from fileheader

update fileheader

update distances

☐ use z-coordinates

reload from file

save changes

load from AsciiFile

save on AsciiFile

close

REFLEXW uses the individual **traceheader coordinates** stored with each trace for the further travelttime 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). 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 auto-

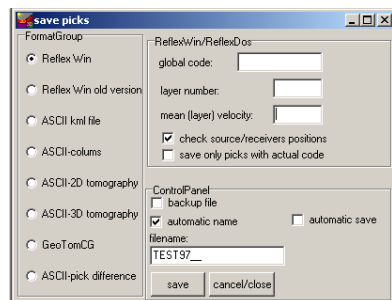
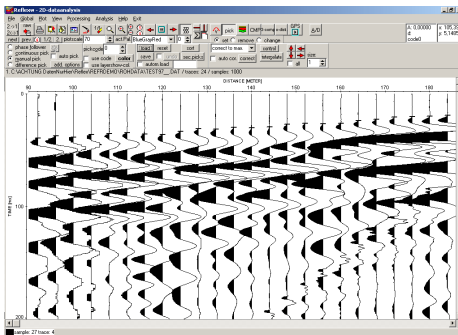
matically shown within a table.

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

- 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).
- 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.

- 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!
- 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 very first arrivals (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).

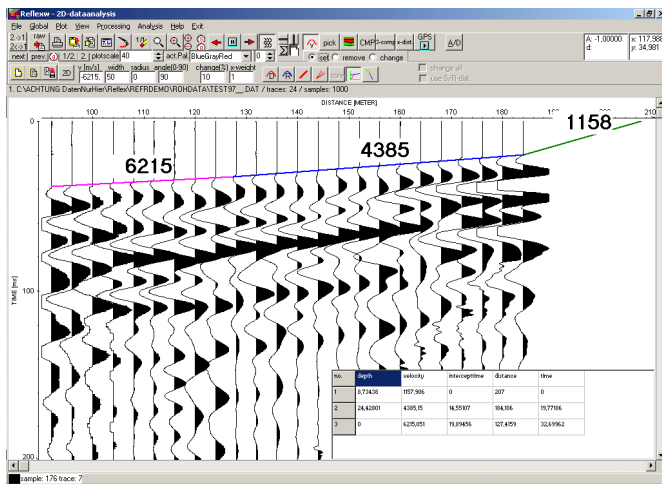


- Do this procedure for all wanted shots. Then the picked traveltimes are ready for interpretation.

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.

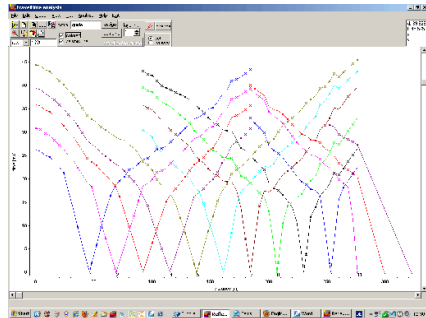
To be considered: If the geophone positions are not equidistant you must activate the plotoption traceheader distances in order to plot the traces at these positions stored within the traceheader distance positions (distance position should be equal rec.x position).



0.7.2 interpretation of the picked traveltimes (done within the module traveltimes analysis 2D)

1. Enter the module **traveltimes 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.



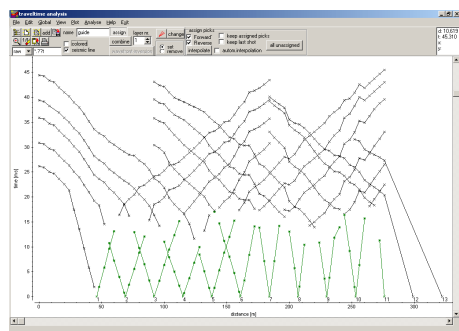
3. Use the option **edit/InsertshotZerotraveltime** in order to insert a zero traveltime at all shot points if this zero traveltime is not still defined. This is useful in order to get a better determination of the velocities of the uppermost layer.

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.



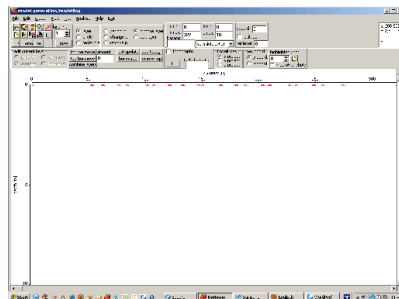
8. Activate the option **combine** in order to do the inversion for the first layer. If no topography is given, do not activate the topography in the CombinePanelLayer1.

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.

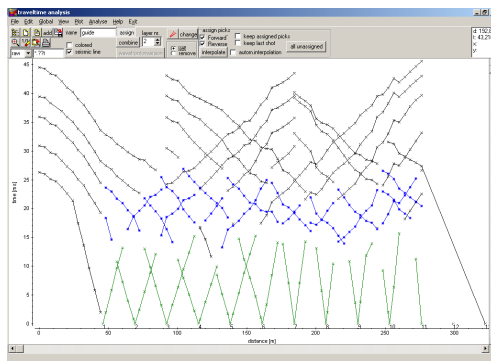
13. Increase the **layer-no.** to 2.

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 traveltimes 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.

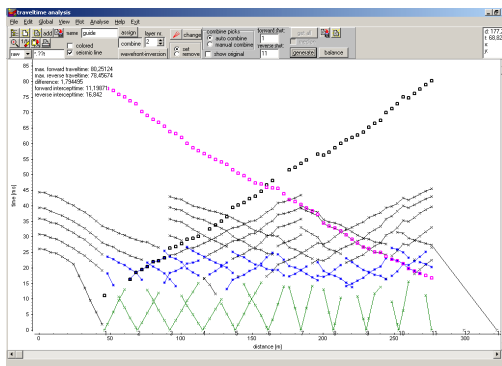


18. Activate the option **generate** when the option autocombine is chosen - a complete combined traveltimetable (forward and reverse) is generated and the total forward and reverse traveltimes are shown.

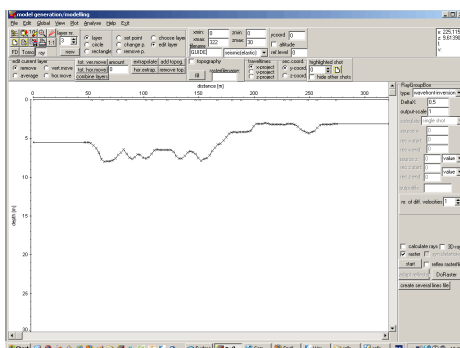
19. If the total forward and reverse traveltimes differ significantly (e.g. more than 5 ms) activate the option **balance** in order to balance the forward and reverse traveltime branches. A large difference can be have difference causes:

- 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 traveltimes 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.

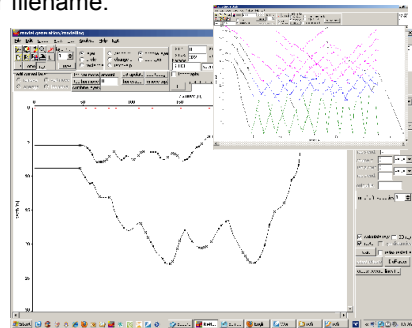
25. Extrapolate the inverted boundary to the model borders (option hor.extrap. or extrapolate) and do some changes of the model if you want to - for example: smoothing is often useful and sometimes some artefacts at the outer borders occur which must be eliminated.

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 0.7.3.



0.7.3 Forward raytracing to refine the model (done within the module model generation/ modelling)

The ray tracing modelling tool allows the traveltimes 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.

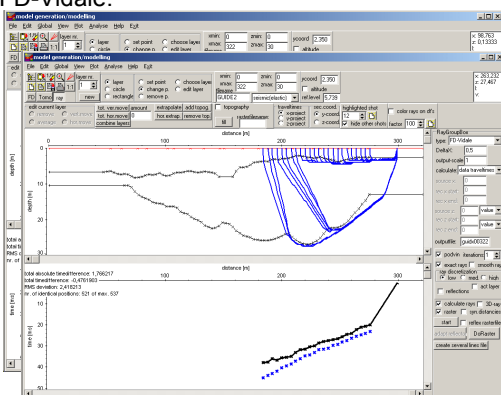
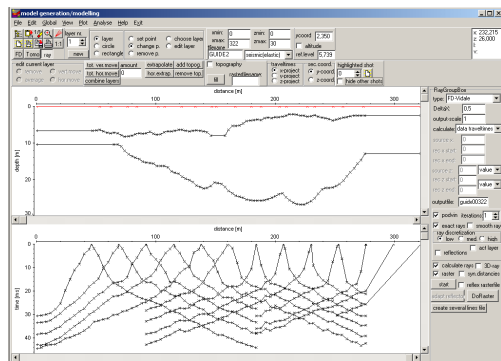
6. Now the ray tracing parameters have to be chosen:

- 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.

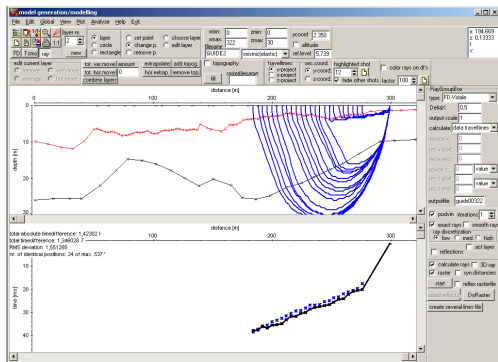
- enter the **output-scale**, e.g. 1.
- enter the **calculate type** - in this case **data traveltimes** because we want to simulate all loaded observed traveltimes.
- enter the **outputfile** name.

7. **Start** the raytracing.

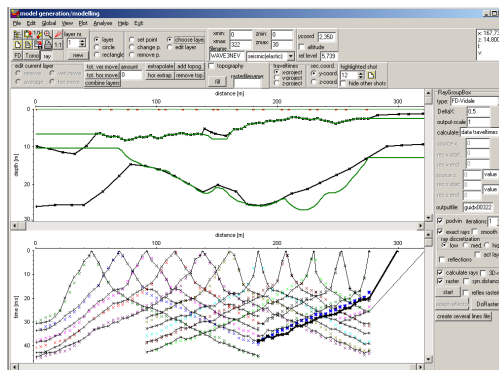


8. At the end of the raytracing the calculated traveltimes are shown in the lower picture in addition to the observed data. Depending on which view option is checked, the calculated rays are shown in addition in the upper picture. Now you may check for the mean traveltime difference using the option **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.



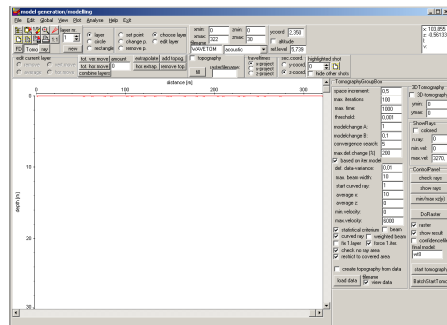
0.7.4. Refraction-tomography: A second kind of interpreting the data

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.

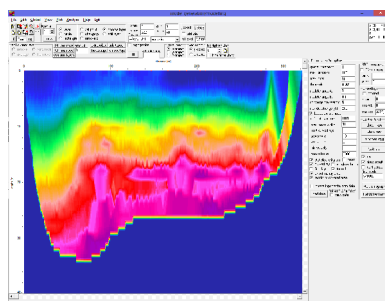
- Load the data (ASCII-data with the extension tom) using the option **load data**. If the data are only available as pck file use the option export to Ascii within the traveltme analysis module in order to generate a tom file.

The program automatically controls, whether the data is in 2D- or in 3D-format.

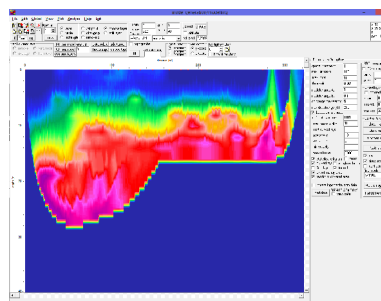
- Enter the wanted **space increment** (equal in x- and z-direction, we used 1 m). Normally this increment should be small enough in order to allow small scale variations with depth. It should be significantly smaller than the receiver increment.

- 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 even 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 shotpoint 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



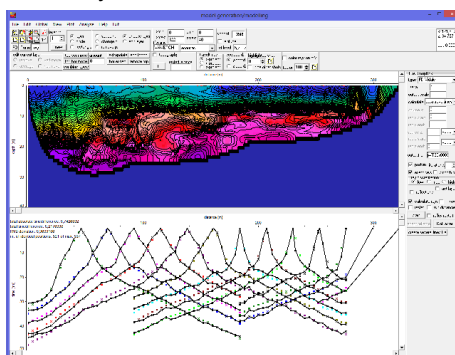
v = 300 m/s

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

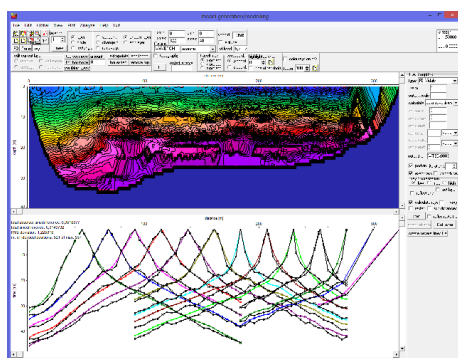
increment used for the tomography (1 m in our case).

- 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 raster 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.



$v = 300 \text{ m/s}$



$v = 800 \text{ m/s}$

Both resulting models show on their right edge ($x > 275 \text{ m}$), 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.

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 too high and the velocity gradient is sufficiently strong, so that great velocity steps are possible. In addition, the gridding increment should not be too small to avoid artefacts.

In contrast to the model resulting from the wavefront-inversion, a shallow low velocity zone appears in the region $x = 150 \text{ m}$ to $x = 280 \text{ m}$ 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 $v = 300 \text{ m/s}$ strongly suggests, that the data in the region $x = 150 \text{ m}$ to $x = 280 \text{ m}$ should be interpreted as an anthro-

pogenic filling, i.e. an area with a strong vertical velocity gradient ($v \approx 500 \text{ m/s}$ up to $v \approx 1000 \text{ m/s}$), instead of a rising boundary with little velocities ($v \approx 400 \text{ m/s}$ up to $v \approx 500 \text{ m/s}$), as the wavefront-inversion model suggests.

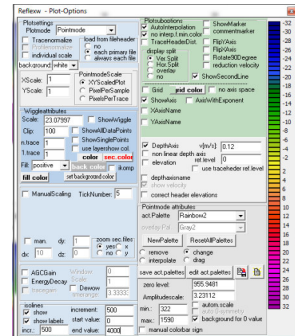
Presentation of the tomographic result:

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:

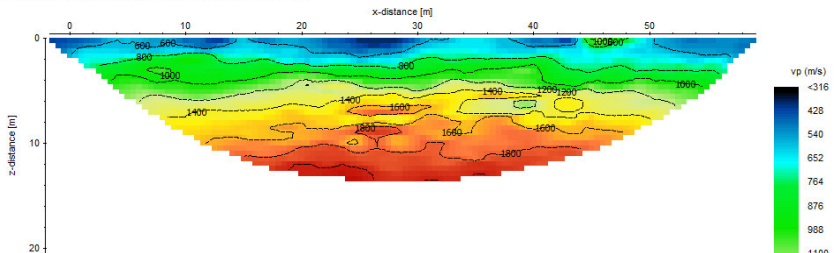
- expand (e.g. using 4 in both directions, set the option keep 0 values to 1)
- median xy-filter (e.g. using 8 in both directions)

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

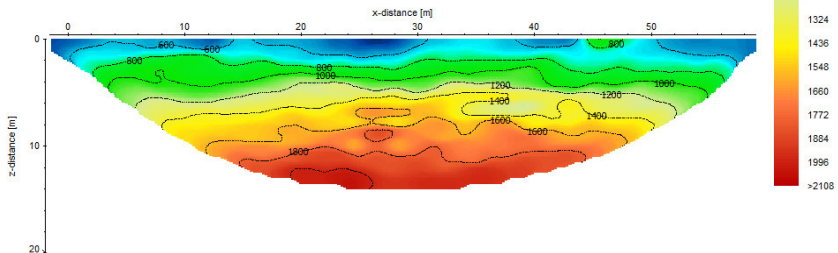
The plotoptions have been set in the following way:
activated option AutoInterpolation and no interp.f.min.color
activated option background for 0 value
Activated options for isolines as shown within the box isolines (see image on the right)



1. F:\data\vis\PROCDATA\Tomo_05_100.DAT / traces: 122 / samples: 41



2. F:\data\vis\PROCDATA\TOMO_05_100.01T / traces: 485 / samples: 161



0.7.5. 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.

Manual change of the wavefront inversion model under consideration of the tomographic results

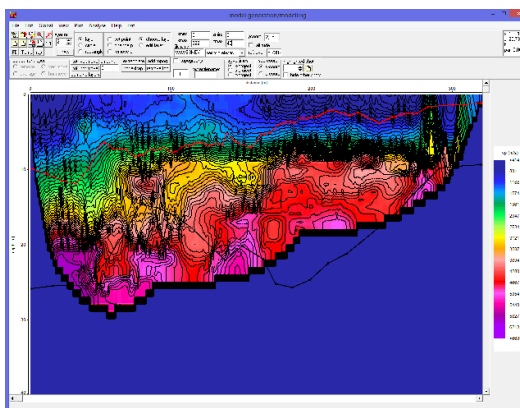
In a first step it is often helpful to display both models together:

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

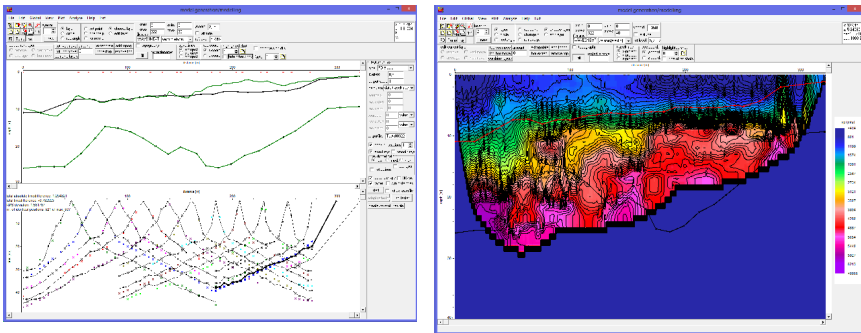
taking into account the tomographic model would lead to an interpretation, which could result in serious consequences for buildings potentially constructed on top of this area! Not until with the help of the tomography model with its strong velocity gradients this near-surface region can be interpreted 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 left 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.

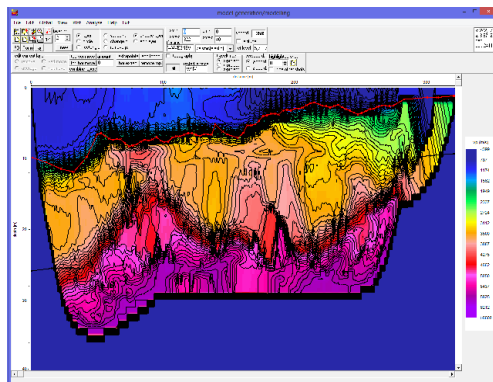
As can be seen in the right figure, the first layer boundary of the refined wavefront-inversion model fits the zone of the narrow isolines of the refraction-tomography model, i.e. the zone with very strong gradients, very well now. So, the refined wavefront-inversion model does map the anthropogenic filling as well.



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 (right figure).

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 lay away along the predetermined layer boundary! Nevertheless, the resulting model – as the tomographic model with a homogenous starting model shown above – contains in the region $x = 150 \text{ m}$ to $x = 280 \text{ m}$ 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.

0.7.6 Topography

The wavefront-inversion and the refraction-tomography do not automatically take into account the topography of a profile regarding its 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 $\alpha = 10^\circ$, the velocity of the bedrock is $v_b = 500 \text{ m/s}$ 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 $v_a = 508 \text{ m/s}$ ($d_g = x / \cos \alpha$ and hence $v_a = v_b / \cos \alpha$, $1/\cos 10^\circ = 1,015$), 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:

$\alpha = 25^\circ$, $1/\cos 25^\circ = 1,103$, $v_b = 500 \text{ m/s}$: $v_a = 551 \text{ m/s}$, $I > 10\%$.

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 traveltimes is already given as **xz-coordinates**, e.g. using GPS no redefinition is necessary and you can skip this chapter.

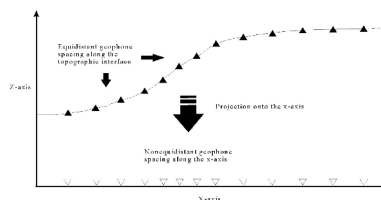
If the data have been acquired along the **topographic interface** and the z-values along the acquisition line have been acquired independently, e.g. using GPS first the geometries of the shots and the receivers must be redefined, whereby the following preconditions must be fulfilled:

- 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, $x = 2$ m, $x = 4$ 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.



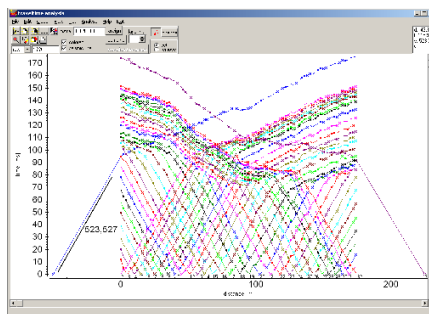
With the help of this ASCII-file the program automatically determines the positions of all shots and receivers on the given topography and calculates the x- and z-projections of these positions: The geometries of the shots and receivers are redefined and can be stored using a new file name. This redefinition 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 its 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.

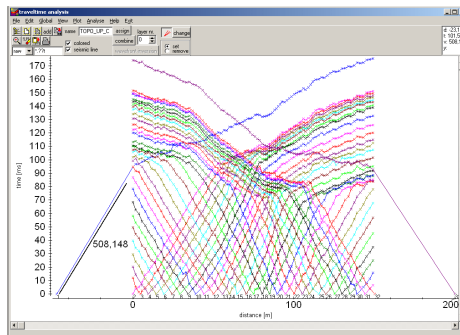
In order to display the data in 'refraction mode' (i.e. the time axis faces upwards) please activate the plot option **FlipYAxis**. If the option **colored** is activated, every shot is displayed in a certain color. This may help to get a first overview of the picked traveltimes.



3. **Start** 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 traveltimes 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.

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.

“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.

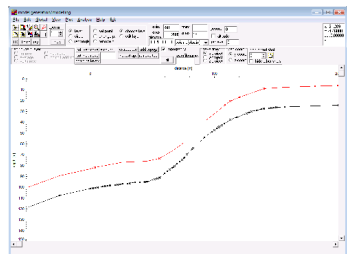
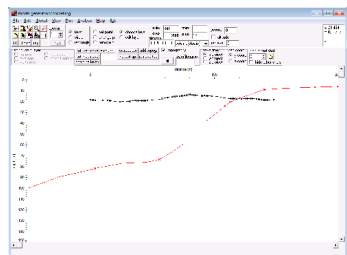
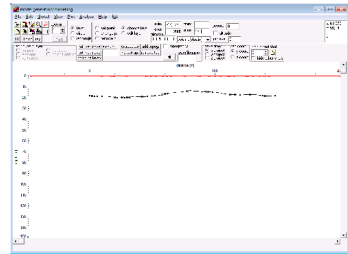
2. **Select the 1. layer and use the option import (x,z)** within the Input of model parameters window to import the topographic xz-values from an ASCII-file. The ASCII-file may either only contain x- (=distance) and z-coordinates or x-, y- and z-coordinates. In the second case the distance along the 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.)



"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.

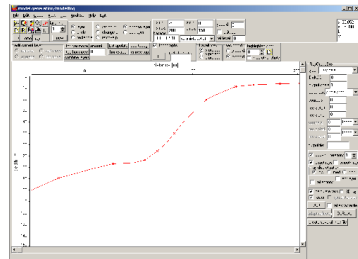
3. Activate the option **assign** in order to assign the wanted traveltimes to layer 1. (It is recommended to deactivate the option **colored** now.)

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.

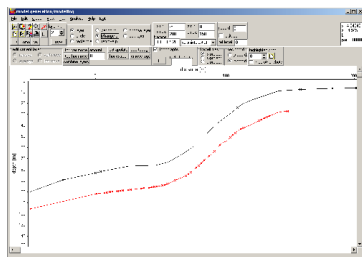
5. Activate the option **combine** in order to do the inversion for the first layer. (To be considered: The inversion for the first layer only consists of the determination of the velocity.)

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.

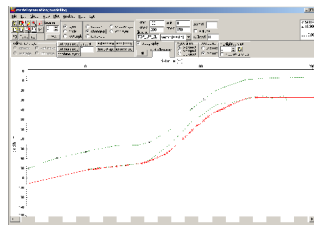
10. Close the modelling window.

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.

0.8 Introduction to the processing of seismic reflection data within REFLEXW

In the following the complete processing of seismic reflection data is described including import of the seismic data (chap. 0.8.1), crosscorrelation (optional for vibration data, chap.0.8.2), setting the geometry (chap. 0.8.3), filtering the shot data (chap. 0.8.4), performing the velocity analysis and stacking (chap. V). Please use in addition to this user's guide the handbook and the online help. The guide describes both the processing of "normal" shot data and of horizontal vibration data. The processing of horizontal vibration data needs some more steps which are highlighted by red.

0.8.1 Import the data (done within the module 2D-dataanalysis)

1. enter the module **2D-dataanalysis**
2. activate the option **file/import**
3. choose the following options within the **import** menu:

data type: several shots

increment: average receiver increment (this increment is used for the equidistant display of the data - the stacking is done based on the traceheadercoordinates which may also be nonequidistant - see also chapter 0.8.2 setting the geometry).

outputformat: new 32 bit floating point (To be considered for SEG2 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.

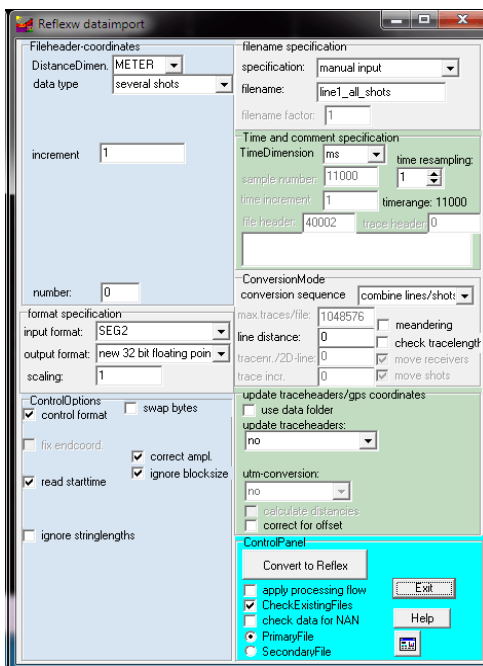
filename specification: manual input

filename: any name (e.g. line1_all_shots)

TimeDimension: ms

conversion sequence: combine lines/shots

line distance: no meaning, may be set to increment



number: no meaning, may be set to 0

The following **plot options** may be set for example (the option may be activated within the import menu (the speed button below the help option)):

Plotmode: Wigglemode

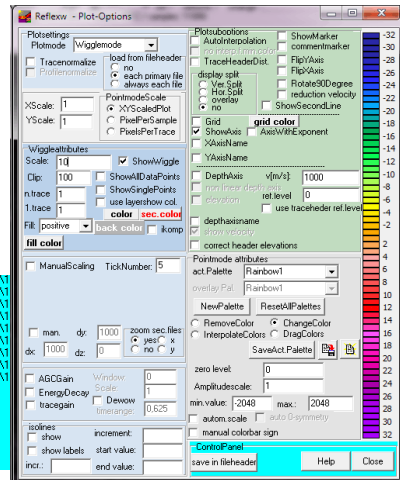
XYScaledPlot activated

Scale: setting depends on the amplitude values of the data. The amplitude values multiplied with scale gives the wiggles size in pixels. The value must be adapted to the original data values.

Optionally you also may activate the option Tracenormalize. Then the option Scale directly gives you the wigglesize in pixels.

4. activate the option **Convert To Reflex**: the wanted shots must be chosen and they are automatically combined into one single datafile containing all the shots.

```
43: inputfile: G:\data\seismic_reflection_vibro_profile1\asca1
44: inputfile: G:\data\seismic_reflection_vibro_profile1\asca1
45: inputfile: G:\data\seismic_reflection_vibro_profile1\asca1
46: inputfile: G:\data\seismic_reflection_vibro_profile1\asca1
47: inputfile: G:\data\seismic_reflection_vibro_profile1\asca1
48: inputfile: G:\data\seismic_reflection_vibro_profile1\asca1
49: inputfile: G:\data\seismic_reflection_vibro_profile1\asca1
50: inputfile: G:\data\seismic_reflection_vibro_profile1\asca1
51: inputfile: G:\data\seismic_reflection_vibro_profile1\asca1
outputfile: G:\data\seismic_reflection_vibro_profile1\VP00DATA\LINE1_ALL_SHOT_PLUS.DAT
traces: 4936 - traceincrement: 1.0000
samples: 11000 - timeincrement: 1.0000
```

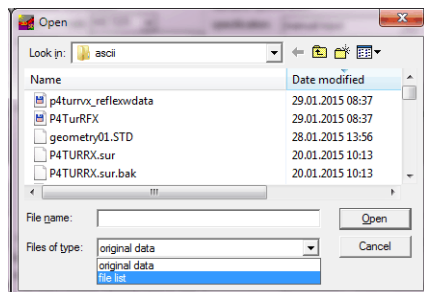


There are 2 different possibilities of choosing the datafiles:

a.. choose the datafiles from the openfile dialog (multiple choice using the shift or str-key) : all choosen original data files are sorted with ascending order of the filenames. Problems may occur if the files don't have the same length (e.g. file1, file2,..., file11). In this case a warning message appears. In order to avoid this the filenames should be chosen in such a way: file001, file002, ... file125.

b. open an external ASCII-filename with the extension ".lst": the external filename contains all wanted datafiles in an arbitrary order (one row contains one filename). The datafiles must be stored under the same path like the ASCII-filename. Example:

```
TEST_02.sg2
TEST_01.sg2
TEST_03.sg2
TEST_04.sg2
```

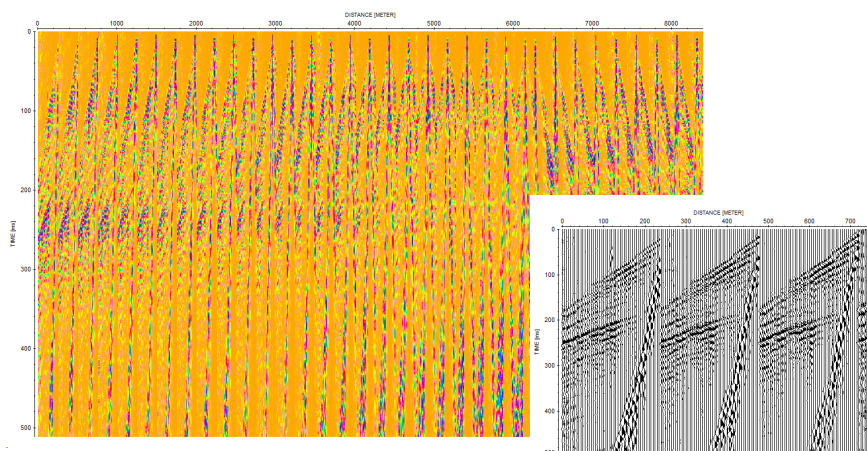


The horizontal vibrator may have two different excitation directions (plus and minus). The data of these different excitation directions must be imported separately.

After having chosen all the wanted datafiles the combined datafile including all

the shots is displayed using the actual plot settings. Because of the normally huge number of traces and the chosen plotmode (Wigglemode, XYScaledPlot) the screen display resolution may be too small to plot the data correctly (XScale too small). In this case choose the Zoom-option in order to only display a small part of the data in x-direction. The plot option point mode allows y higher resolution but it is not so useful for the further processing, e.g velocity analysis.

NOTE: The standard display within the 2D-dataanalysis uses the product of the given traceincrement and the actual tracenummer for the axis display. Therefore this display does not represent the correct coordinates for the combined shot data and you may ignore them. But you don't have to worry about them because the subsequent sorting and stacking is always done based on the traceheader coordinates (see chap. 0.8.3). The coordinates of resulting stacked section (see chap. V) however are displayed correctly within the standard display.

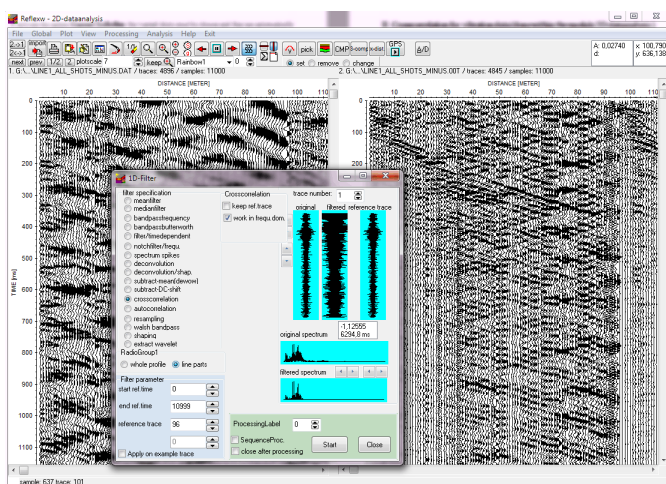


0.8.2. Crosscorrelation for vibration data (done within the module 2D-dataanalysis)

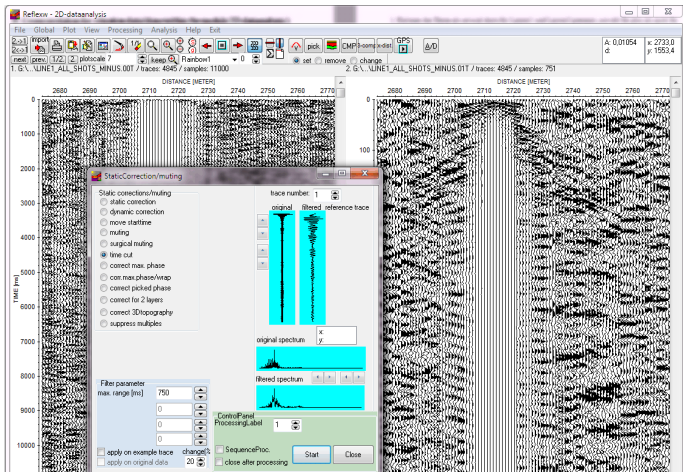
If the data have been acquired using a vibrator the data must be first crosscorrelated with the sweep signal before these can be interpreted. The vibrosource emits a long sinusoidal signal with varying frequency over several seconds (in this case 10 s). The recorded signal is a mixture of many time-delayed copies of the signal. Cross-correlation is then used as a trace compression using the original signal and the observed data to unravel all of this. The processing flow:

1. Load the file containing all combined shots
2. Enter processing/1D-filter
3. Activate crosscorrelation
4. Enter the start and end reference time (normally the total time range, also by default)
5. Enter the position of the reference trace, in this case channel no. 96 represents the sweep signal
6. Activate the option lineparts if the datafile consists of several individual shot datasets
7. Deactivate the option keep ref.trace because the sweep signal is no longer necessary
8. The method works both in time and frequency domain. Both methods produce similar results but the frequency domain method is faster.

The resulting crosscorrelated traces now are comparable to “normal” shot data (e.g. using a hammer or an explosive source).



It is recommended to restrict the timerange, in this case to 750 ms, using the option timecut under processing/static correction



0.8.3. setting the geometry

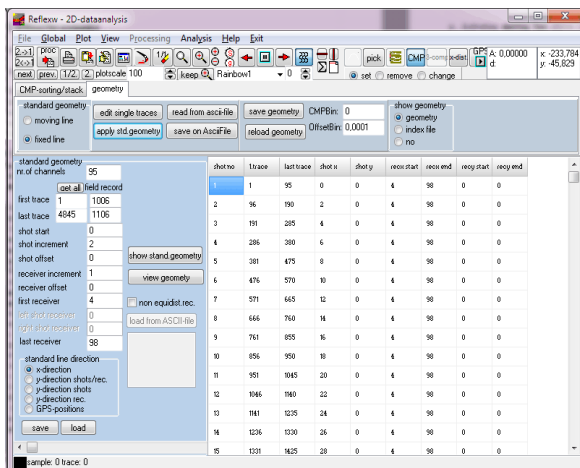
1. activate the option **CMP** within the 2D-dataanalysis
2. activate the option **geometry**

If the geometries have been already entered during the data acquisition and REFLEXW allows to automatically take over these geometries the geometry is shown in a tabella with shot number, shot position and receiver line position. If the geometry is okay you may leave the geometry settings by deactivating the option CMP and go on with chapter 0.8.4

If the geometry is not correct you must proceed with the following steps:

3. There are different possibilities to define the geometry. The most convenient way is to use a standard geometry (option **moving line** and option **fixed line**) for a selectable number of traces. Apart from the standard geometry it is also possible to edit the geometry of single traces (option **edit single traces**) or to load the geometry from an ASCII-file (option **read from ASCII-file**).

a. Activating **fixed line** allows you to define the geometry for a fixed geophone line for different shot points. An example of the input parameters is shown on the next figure. Nr.of channels is the number of channels per shot. First and last trace define the range on the profile for which the standard geometry should be valid. This gives you the opportunity to declare standard geometries for different ranges of the profile. Instead of the first and last trace it is also possible to define the range using the original field record numbers. Shotstart is the position of the shot of the first ensemble, shot increment is the distance between successive shots and shot offset is the distance between a shot and the profile in perpendicular direction. Receiver increment is the distance between successive receivers and receiver offset is the distance between a geophone and the profile in perpendicular direction. First receiver and last receiver define the position of the geophones with respect to the shot. These are the absolute coordinates in contrast to moving line where relative coordinates (to the shot) are used.



b.. Activating **moving line** allows you to define the geometry for a geophone line moving with the shots. An example of the input parameters is shown on the right figure. Nr.of channels is the number of channels per shot. First and last trace define the range on the profile for which the standard geometry should be valid. This gives you the opportunity to declare standard geometries for different ranges of the profile. Instead of the first and last trace it is also possible to define the range using the original field record numbers. Shotstart is the position of the shot of the first ensemble, shot increment is the distance between successive shots and shot offset is the distance between a shot and the profile in perpendicular direction. Receiver increment is the distance between successive receivers and receiver offset is the distance between a geophone and the profile in perpendicular direction. First receiver, left shot receiver, right shot receiver and last receiver define the position of the geophones with respect to the shot.

standard geometry	
nr.of channels	48
field record	
first trace	1
last trace	1488
shot start	100
shot increment	2
shot offset	0
receiver increment	2
receiver offset	0
first receiver	-48
left shot receiver	-2
right shot receiver	2
last receiver	48
<input type="checkbox"/> y-line direction	
<input type="button" value="save"/> <input type="button" value="load"/>	

c. The option **read from ascii-file** allows you to read the geometry from an ASCII-file. The ASCII file must exist under the path ASCII and must have the extension DST. Each line of the ASCII file contains the following 6 informations:
tracenumber distance Shot-X-Pos Shot-Y-Pos receiver-X-Pos receiver-YPos
 Defining the tracenumber gives you the opportunity to read the geometry only for a distinct part of the data.

d. The option **edit single traces** allows you to change the geometry of each trace individually. When activating this option the trace header Edit menu will be opened where you can edit the shot and receiver coordinates for each trace separately.

4. If you want to apply a standard geometry (3.a or 3.b) you must activate the option **apply std. geometry** in addition.

5. If necessary enter an additional standard geometry and apply it onto the profile or choose one of the other geometry edit possibilities.

6. If the geometry setting is finished choose the option **save geometry** in order to save the geometry within the traceheader coordinates of the file.

7. Now the file is ready for pre-stack processing and stacking.

0.8.4 pre-stack filtering (done within the module 2D-dataanalysis)

The aim of the pre-stack filtering is to prepare the dataset for the subsequent stacking. Therefore the most important processing steps are:

- energy normalization
- elimination or suppression of the surface waves and other non desired waves

The **energy normalization** should be the first filter step especially if you are using multi-channel filters (e.g. FK-filter) for the suppression of non desired waves.. This filter step is applied to correct for the amplitude effects of wavefront divergence and damping. It is also necessary if a subsequent filter shall be applied for eliminating the surface waves.

The easiest way would be a manual gain function, e.g. manual gain (y). In the case of strong surface waves this gain recovery function is not very suitable. In this case the filter scaled windowgain(x) may be a good choice.

To be considered: Filtering and stacking are always done on the true amplitude data. The option `tracenormalize` within the `plotoptions` is only a `plotoption` and does not effect the filtering/stacking process. Therefore the option `tracenormalize` within the `plotoptions` should be deactivated because otherwise the effect of the energy normalization may not be controlled correctly.

The second filter step consists of the **elimination of the surface waves** and other non desired waves. For that purpose you may distinguish between editing and 2D-filter steps.

Editing may be used for muting (set to zero) special data parts or to delete single traces or trace ranges.

2D-filters may be used to suppress unwanted signals. A FK-filter may be a good choice.

0.8.4.1 prestack filter sequence for data exhibiting surface waves

In the following the processing using a scaled windowgain(x) and a FK-filter is described:

1. load the wanted raw data for which the geometry has already been defined (all shots are stored within one file, see chap. I and II).

2. enter the **first** filter step using the option **Processing/gain**.

3. The **Gain window** appears. Enter the following parameters or options:

- activate the option **scaled windowgain(x)**

- enter the filterparameter **start window** and **end window**, the window size should be chosen in such a way that a good energy normalization is fulfilled.

Default values are:

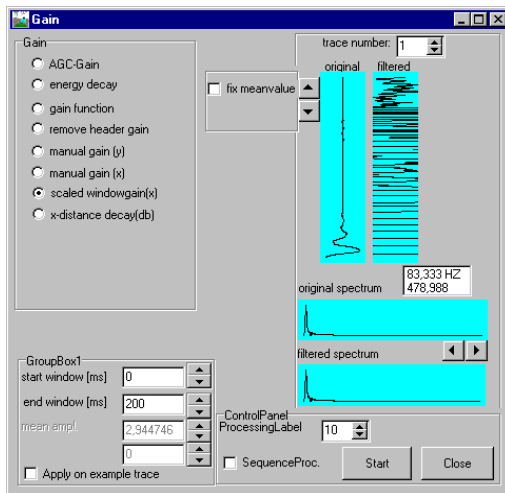
start window: 0

end window: the traveltime of the main reflection at the greatest distance offset.

- enter the wanted

ProcessingLabel

- **start** the filtering



4. The filtered data have been

plotted into the secondary

window (depending on the

settings of the screen splitting

within the plot option menu). Whereas in the original data only the surface waves and the very first arrivals are visible due to the non normalized display the filtered data show clear refracted waves for all distances and some reflections. Now

close the filter window and

you must enter the option

File/ChangeSecondToPri

mary in order to use the

filtered dataset for the next

desired filter step.

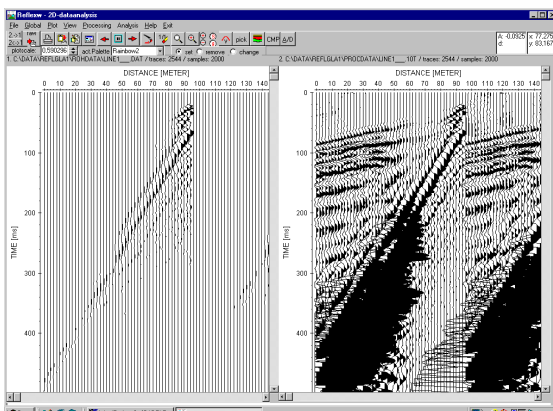
5. Choose the filter which

allows to eliminate or at

least decrease the surface

waves.

Surface waves normally differ from the body waves in two significant characteristics. These



properties involve the frequency and the apparent velocity. Both are often significantly lower than those of the body waves.

If a lower frequency content is inherent a **high pass bandpassfiltering** is the best choice.

5.1 enter **processing/1D-filter** and choose **bandpassfrequency**

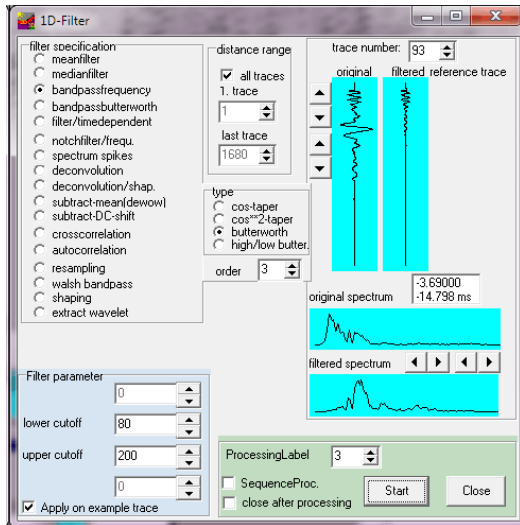
For the tapering the butterworth type of the order 3 is often useful.

Enter the wanted lower and upper cutoff frequency. If the option Apply on example trace the original and the filtered trace as well as the original and the filtered spectrum are displayed.

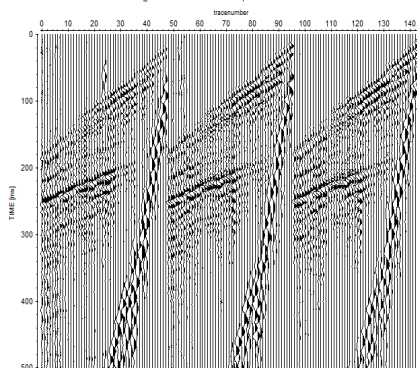
In this case the high pass bandpass filtering allowed a very well elimination of the surface waves - see picture below.

If the bandpass filtering is not sufficient a **fk-filtering** might be useful. Here it must be considered that no spatial aliasing occurs. This may happen if very low velocities are present together with a too high receiver increment. Then after applying the fk-filter some energy of the surface waves will remain.

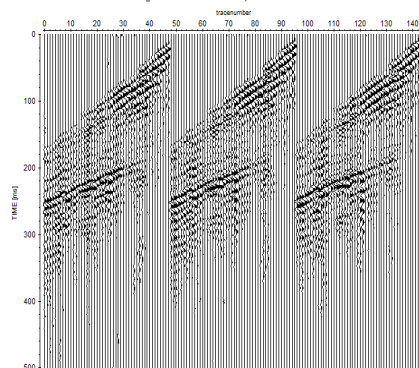
5.2 enter the second filter step using the option **Processing/FK-filter/FK-spectrum**.



1. F:\DATA\REFLMAR\PROCDATA\LINE_ALL.DOT / traces: 1680 / samples: 2048



2. F:\DATA\REFLMAR\PROCDATA\LINE_ALL.DOT / traces: 1680 / samples: 2048

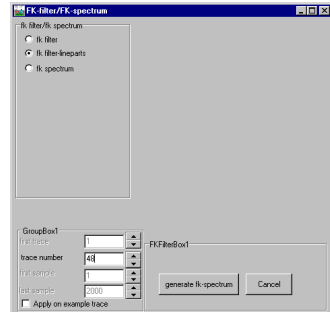


The FK-filter/FK-spectrum window appears. Enter the following parameters or options:

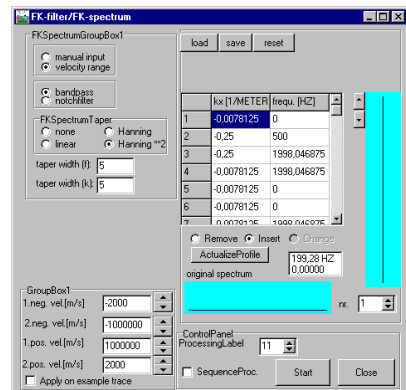
- activate the option **fk filter-lineparts**
- enter the filterparameter **tracenum** (corresponds to the number of traces per shot)
- activate the option **generate fk-spectrum**

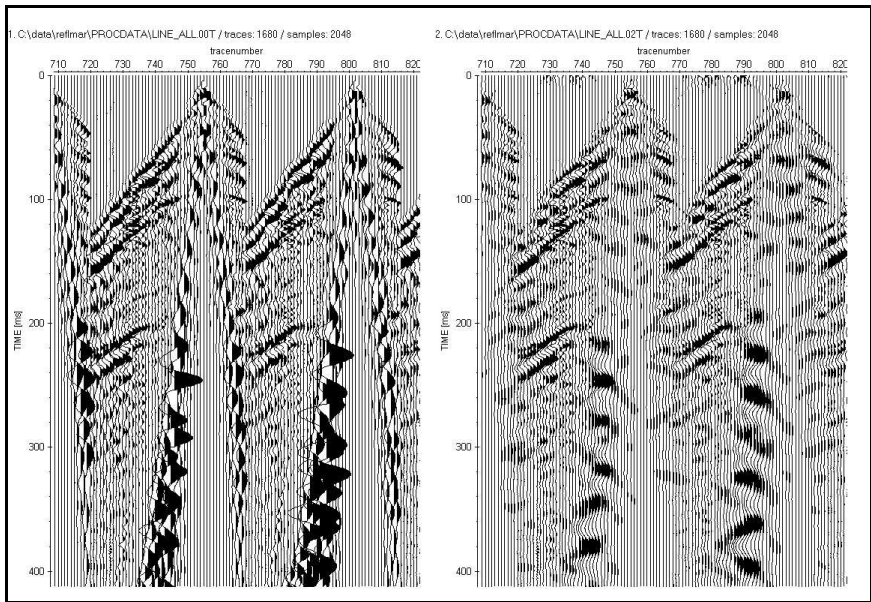
A new FK-filter/FK-spectrum window appears. Enter for example the following parameters or options:

- activate the option **velocity range**
 - activate the option **bandpass**
 - activate the option **Hanning **2**
 - enter 5 for the **taper widths**
 - enter the **velocity fan** for example:
1.neg.vel. to -2000, 2.neg.vel. to -1000000,
1.pos.vel. to 1000000 and 2.pos.vel. to 2000.
- These values may differ significantly from case to case. The velocity fan should be set in such a way that all non desired onsets are suppressed.
- enter the wanted **ProcessingLabel**
 - **start** the fk-filtering



The filtered data have been plotted into the secondary window (depending on the settings of the screen splitting within the plot option menu). The surface waves as well as the first arrivals (refracted waves) are quite well suppressed. If the result is sufficient the pre-processing is finished and you may continue with chapter 0.8.5. If not, some other filter or editing steps must be performed.

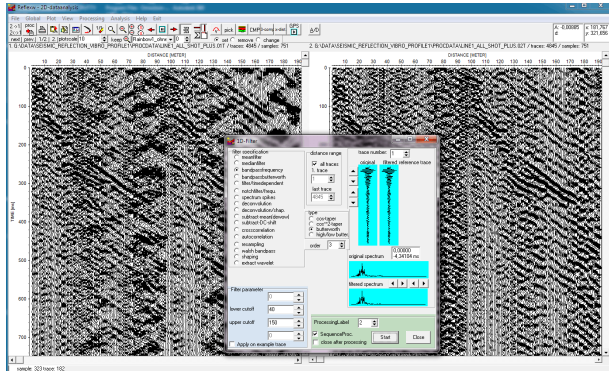




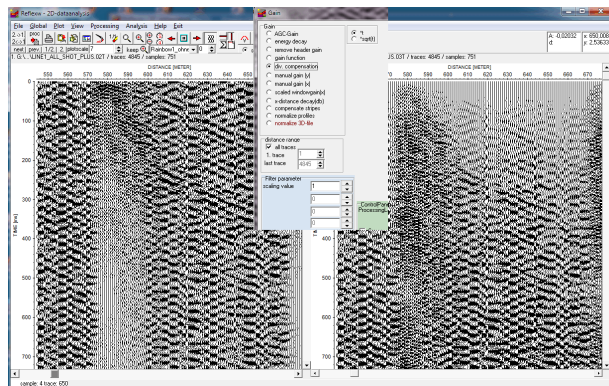
0.8.4.2 prestack filter sequence for horizontal vibration data

1. load the wanted raw data for which a crosscorrelation has been done and the geometry has already been defined (all shots are stored within one file, see chap. I to 0.8.3).

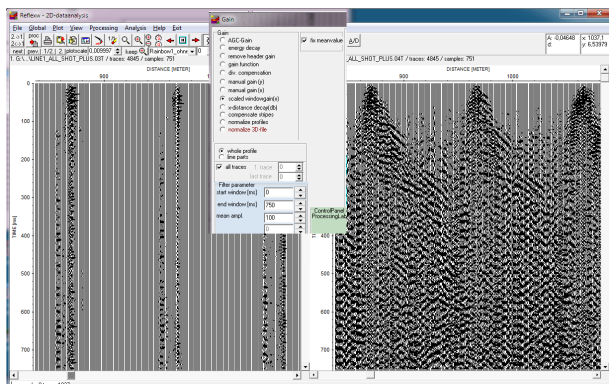
2. enter the first filter **bandpassfrequency** within Processing/1D-filter, in this case a frequency range between 40 and 150 Hz and a butterworth taper of order 3 has been used. The **tracenormalize** plotoption has been activated in order to view the filter effects.



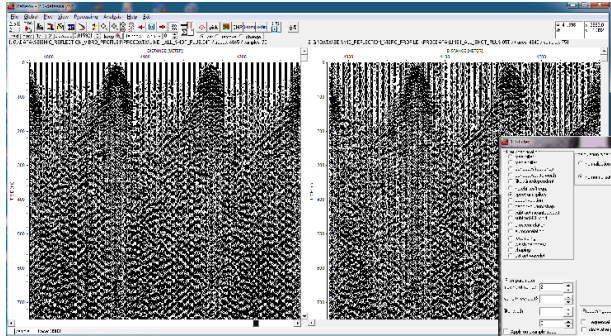
3. Enter the second filter step **div. compensation** under processing/gain which compensates for the geometrical divergence losses



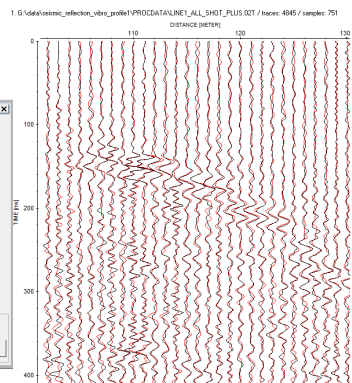
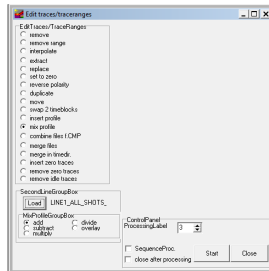
4. Enter the third filter step **scaled windowgain (x)** under processing/gain which facilitates the amplitude equality distribution in x-direction for a y(time)-window to be specified. The **tracenormalize** plotoption has been deactivated in order to view the true amplitude representation.



5. Enter the fourth filter step **spectrum spikes** under processing/1D-filter which is suitable for suppressing nearly mono-frequency noise which is not restricted to a special frequency but may vary from trace to trace.

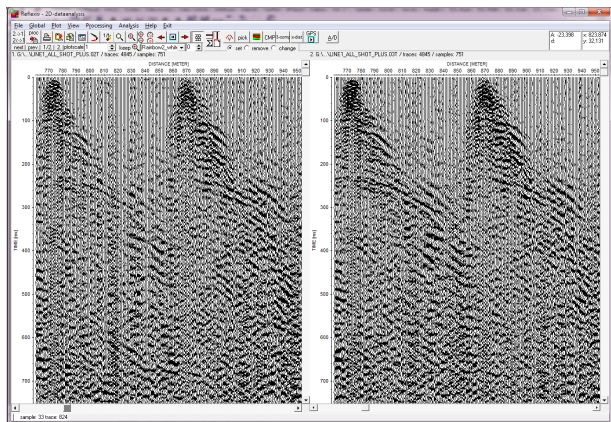


6. The filter steps described above must be applied on the two datasets of different signal excitation (plus, minus). Now the processed data may be combined using the option **mix profile** with suboption **add**. Although the original non crosscorrelated data of course exhibit different signs (plus and minus signal excitation) both datasets must be added because the different signs will be lost during the crosscorrelation process.



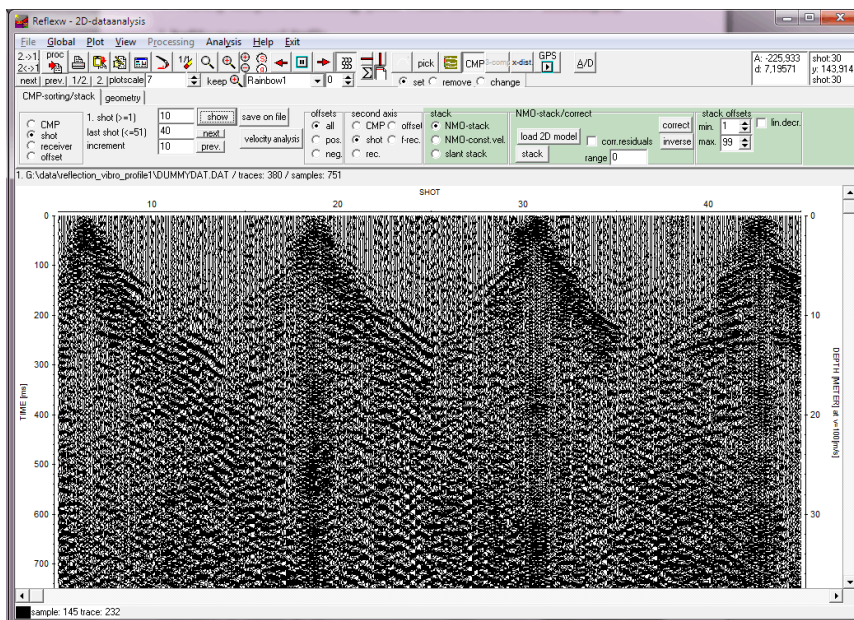
Now the data are ready for stacking (see chap. 0.8.5).

Within this chapter we have discussed two example of a possible pre-processing. It is not possible to give some general rules for the processing because of the huge different problems involved with the datafiles. In any case a gain recovery (energy normalization) and a suppression of the unwanted onsets must be performed.



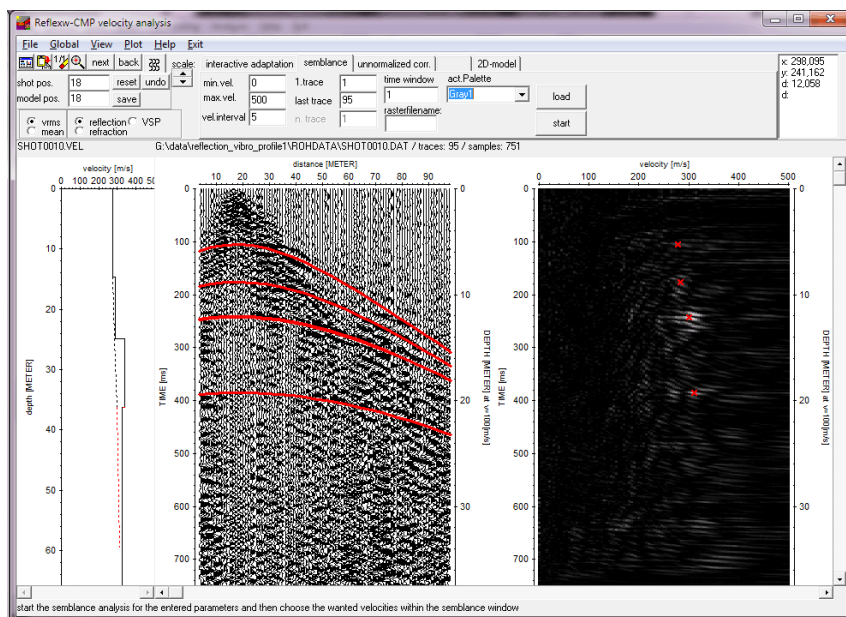
0.8.5 velocity analysis and stacking (done within the module 2D-dataanalysis)

1. **load** the pre-processed datafile
2. activate the option **CMP** within the 2D-dataanalysis
3. click on **CMP-sorting/stack**
4. Choose the sorting option **CMP** or **SHOT**
5. choose the **first** and **last CMP** (or SHOT) and the **increment** for which a velocity analysis shall be done (example 10 for the first shot, 40 for the last shot and 10 for the increment in order to do the velocity analysis for 5 different shots).



6. activate the option **velocity analysis**
7. the mean **traceincrement** for the ensembles is asked for. This increment is used for the equidistant display of the CMP or shot-ensembles.
8. The velocity analysis menu opens with the first CMP(shot)-data ensemble loaded. On the left the actual layer model is shown (by default one single layer included). On the right the CMP (or Shot) is shown.

9. Activate the option **semblance** or **unnormalized corr.** for doing a first estimation of the velocity model.



10. Enter the velocity range and increment and **start** the semblance analysis. The semblance is shown in an additional window.

11. set the left mouse button at the wanted positions within the semblance analysis window in order to create a velocity model. The velocity model on the left is automatically updated.

12. Activating the option **interactive adaptation** allows you to refine the model - for a detailed description of the options see online help.

13. **Save** the model

14. click on **Next** to proceed with the next CMP (shot).

15. The last velocity model remains and you may refine the model using the interactive adaptation or may create a completely new model using the semblance or unnormalized corr. (see point 9).

16. **Save** the velocity model.

17. Repeat step 14 until 16 until the last wanted CMP (shot) is reached

18. Activate the option **2D-model**

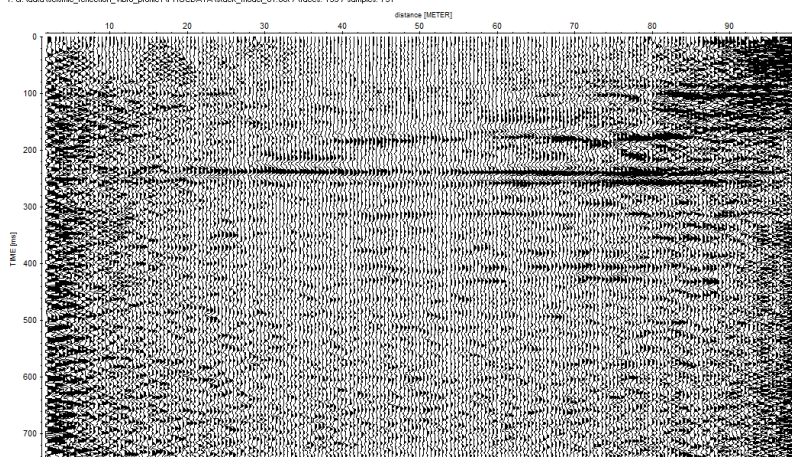
19. **create** a new 2D-model - after having activated the option create you are asked for the wanted 1-dimensional velocity model. Choose the wanted models from the open file dialog (multiple choice using the shift or str-key), enter a filename of the 2D-model and the interpolated 2D-model is plotted into the right window.

20. **Exit** the CMP-velocity analysis menu.

21. Choose **NMO-stack** and load the wanted 2D-model using the option **load 2D-model**.

22. Enter the wanted sorting ensemble (e.g. CMP) and enter the ensemble range for the stacking (e.g.: 1. CMP and last CMP and increment whereby the increment should be set to 1). Start the stacking using the option **stack**. Enter a filename for the stacked data and enter the traceincrement (a calculated value based on the fileheader traceincrement of the original file is given by default). The stacked section is stored under the path procddata using the processing label 0 and is shown in an additional window. To be considered for the stack section: The default value of the trace increment is given by the number of stack traces and the coordinate range. If this mean traceincrement does not equal any increment between successive stack traces the resulting stack traces are obviously not equidistant. This may happen for example if the shot interval is not equidistant for all shots. In this case the program (from version 3.03) gives a warning message and asks if equidistant traces based on the mean traceincrement shall be made from the non equidistant stack section. If this procedure is cancelled the resulting stack section not equidistant and you must activate the plotoption TraceHeaderDistancies (only enabled for the wiggle mode) or you must make the stack section equidistant afterwards.

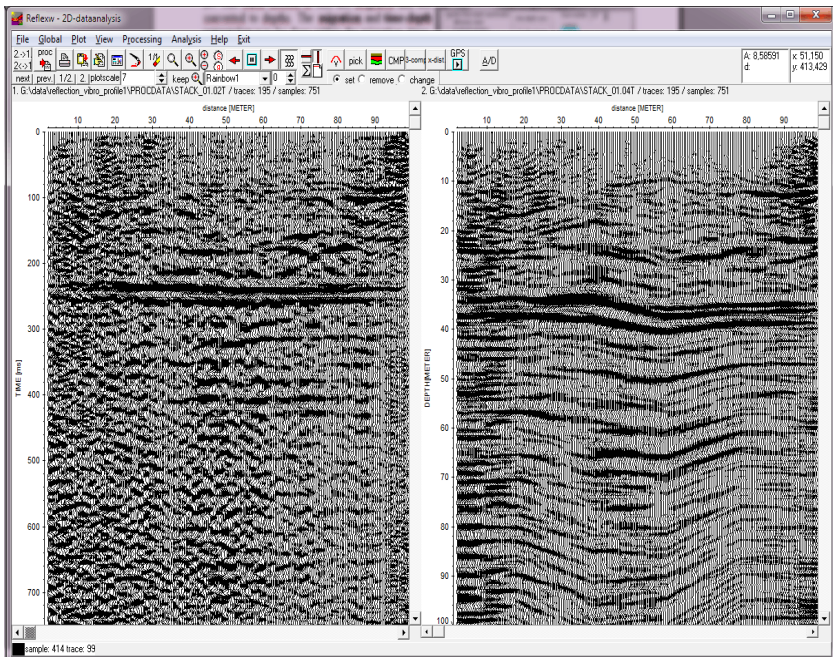
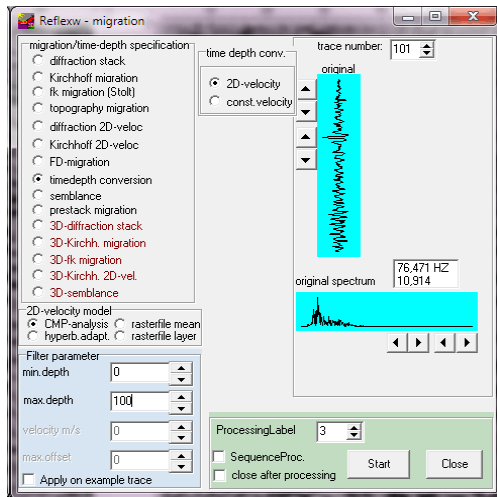
1. G:\data\seismic_reflection_vibro_profile\PROCDATA\stack_model_01.001 / traces: 195 / samples: 751



23. Now the stacking is finished and you may look for it and do some **post-processing** within the 2D-dataanalysis (see also guide filtering). For that

purpose deactivate the option CMP and load the stacked section.

24. The stack section is still not migrated and not converted to depths. The **migration** and **time-depth conversion** may be done within the migration /time-depth conversion menu. The example shows the processing based on the same 2D-velocity model used for the stacking. For the migration the Kirchhoff 2D-veloc. option has been used and for the timedepth conversion the parameters shown on the right. The lower panel shows the original stack section (left) and the section after migration and timedepth conversion (right).



0.8.6 Prestack migration as an alternative to the standard NMO-stacking

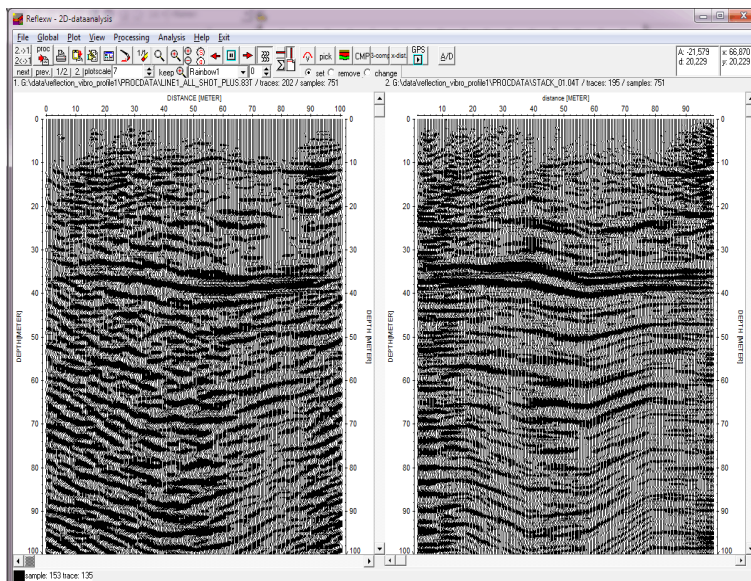
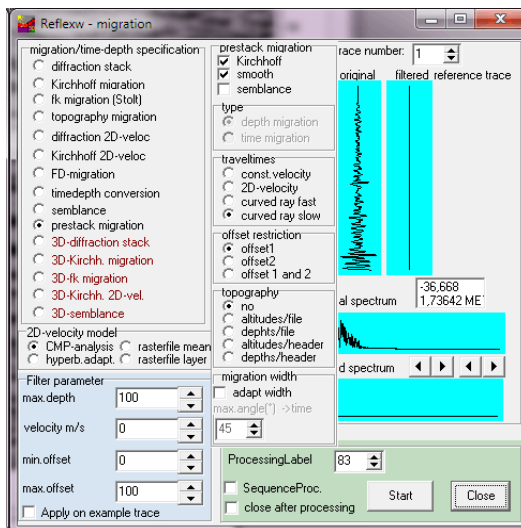
The prestack migration is an alternative possibility for getting a stacked section. The resulting section is automatically migrated and converted to depth.

1. load the pre-processed datafile

2. Enter the processing option prestack migration under processing/migration

3. Enter the 2D-velocity model and the max. depth as well as the min./max. offsets. If a 2D-velocity shall be the same of the migratio use curved ray slow.

4. The lower panel shows the comparison of the prestack migrated section and the NMO-stacked section including migration and timedepth conversion.



0.9 Introduction to the 3D-datainterpretation within REFLEXW

In the following the 3D-datainterpretation of REFLEXW is described including the generation of a 3D-dataset (Chap. 0.9.1), the processing of 3D-datafiles (chap. 0.9.2) and the interpretation of 3D-datafiles (chap. 0.9.3). Please use in addition to this guide the instructions within the corresponding chapters of this manual and the online help.

0.9.1. generation of a 3D-dataset

A **3D-data file** is a single REFLEX formatted file which consists of several equidistant 2D-lines sequentially stored. All traces belonging to one 2D-line have the same ensemble-number which is stored within the REFLEXW traceheader. The direction (x or y) within the fileheader determines the direction of the 2D-lines.

Depending on the original data there are several ways to generate a 3D-file in ReflexW format:

generate 3D-file without an interpolation:

Precondition is that the original data have been acquired along **parallel equidistant 2D-lines**. If this is true the 3D-file may be generated

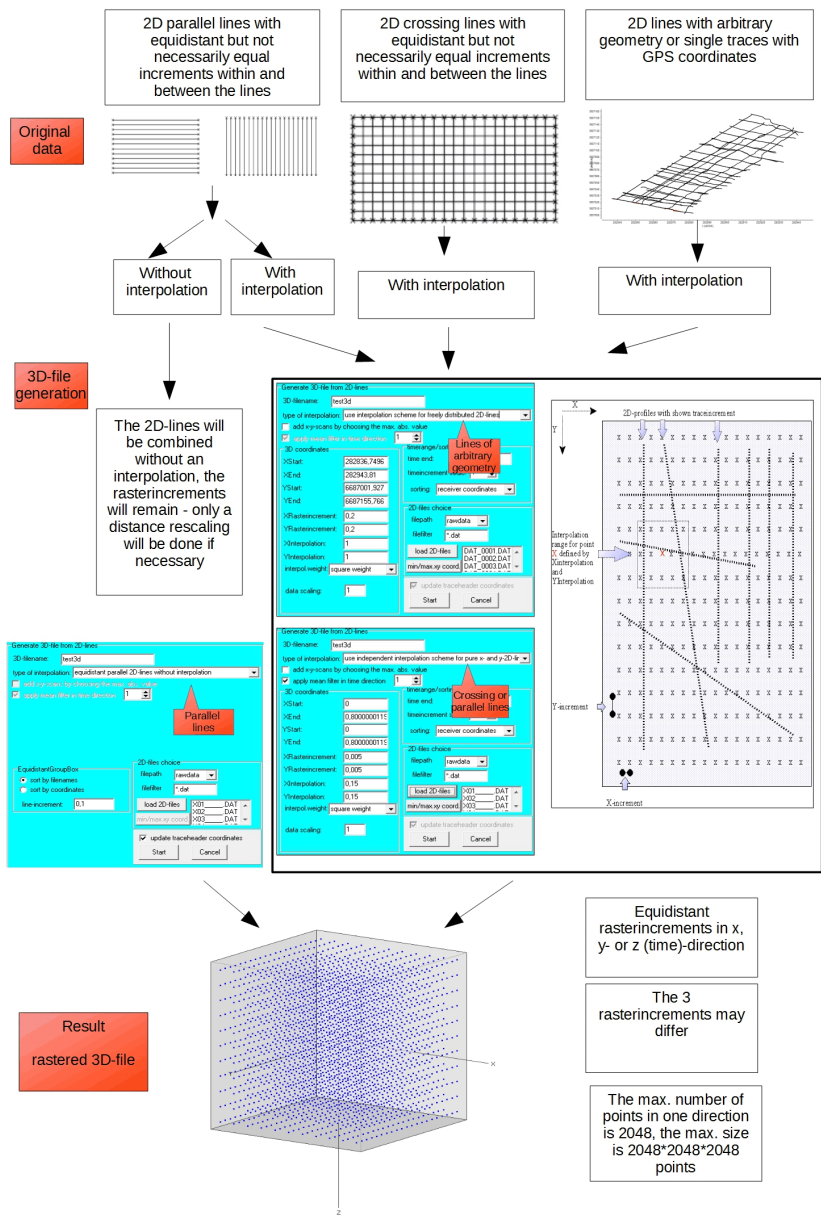
- from original 2D- or 3D-data during the import (chap. 0.9.1.1.1). In this case the complete dataprocessing is done for the 3D-datafile (see chap. 0.9.2.).
- from REFLEXW formatted 2D-lines (chap. 0.9.1.1.2). The 2D-lines may be raw data (then the processing is done for the 3D-datafile) or already processed data. There are no general rules whether the processing shall be done for the 3D-datafile or for the individual 2D-lines.

The rasterincrements of the resulting 3D-datafile will not be changed.

generate 3D-file with interpolation:

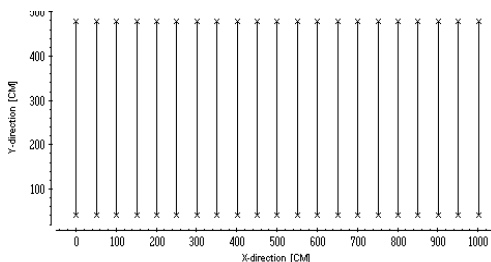
The 3D-datafile may be constructed from Reflexw formatted **parallel 2D lines**, **crossing 2D lines** (chap. 0.9.1.1.2) or from **freely distributed lines** using e.g. GPS based coordinates (chap. 0.9.1.1.3). The rasterincrements of the resulting 3D-datafile may be freely entered.

In both cases, the data points of the resulting 3D-file have fixed increments in x-, y- and time-direction, respectively, and the max. size of the 3D-file is limited to 2048 points in each direction. The resulting 3D-datafile has the same REFLEXW format like a 2D-datafile (the ensemble number within the traceheader controls the sequential storing) and it therefore may be processed within the 2D-dataanalysis module (chap. 0.9.2) whereby the interpretation must be done within the 3D-datainterpretation (chap. 0.9.3).



0.9.1.1 Generating a 3D-file without interpolation

A 3D-file without interpolation may be generated from original 2D- or 3D-data during the import (chap. 0.9.1.1.1) or from REFLEXW formatted 2D-lines (chap. 0.9.1.1.2). In both cases the original data are parallel 2D-lines with equidistant profile increment orientated either in x- or y-direction.



0.9.1.1.1 Generating a 3D-file from during the import from parallel 2D-lines

The original data are parallel 2D-lines which are either stored as separate 2D-files or stored within one single file including all 2D-lines.

A. original parallel 2D-lines stored as separate files

The precondition is that the original data are:

- parallel 2D-lines with a fixed profileincrement (increment between the different 2D-lines) and identical startcoordinate
- fixed traceincrement
- fixed timeincrement
- same number of points per trace (identical timerange)
- same number of traces per 2D-line (identical distancerange)

Then a 3D-datafile may be easily constructed during the import by simply storing sequentially the different 2D-lines within one file. The size of the 3D-cube is determined from the number of 2D-lines, the number of traces per 2D-line and the number of points in timedirection. Example: 21 different 2D-lines with 200 traces per 2D-line (profiledirection: y) and 256 points per trace: xpoints: 21, ypoints: 200, zpoints (timepoints): 256.

In the following the individual steps for creating such a 3D-file are described in detail:

1. enter the module **2D-dataanalysis**
2. activate the option **file/import**
3. choose the following options within the **import menu**:

data type: const. offset (version 2) or 3D-const.offset (from version 3)

ProfileDirection: direction of the original 2D-lines (enter either x or y)

ProfileConstant: constant coordinate direction of the original 2D-lines (enter either y or x depending on the chosen profile direction)

xstart: start coordinate in x-direction

ystart: start coordinate in y-direction

xend, yend, zstart, zend: not necessary to input

outputformat: 16 bit integer (e.g. for GPR-data) or 32 bit floating point for a higher data resolution (e.g. for seismic data).

To be considered for SEG2 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.

filename specification: manual input

filename: any name (e.g. line_3d)

TimeDimension: ns (GPR data) or ms (seismic data)

conversion sequence: combine lines/shots

linedistance: enter the distance between the individual parallel 2D-lines

4. activate the option **Convert To Reflex**: the wanted shots must be chosen and they are automatically combined into one single datafile containing all the shots.

There are 2 different possibilities of choosing the datafiles:

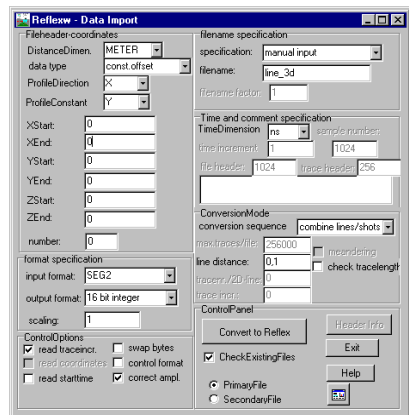
a.. choose the datafiles from the openfile dialog (multiple choice using the shift or str-key) : all chosen original data files are sorted with ascending order. Problems may occur if the files don't have the same length (e.g. file1, file2,..., file11). In this case a warning message appears.

b. open an external ASCII-filelist with the extension ".lst": the external filelist contains all wanted datafiles in an arbitrary order (one row contains one filename). The datafiles must be stored under the same path like the ASCII-filelist. Example:

TEST__02.sg2

TEST__01.sg2

After having chosen all the wanted datafiles the combined 3D-datafile is displayed using the current plot settings.



B. original parallel 2D-lines stored as one single 3D-datafile

In this case it is assumed that the original profile is a 3D-file consisting of parallel 2D-lines with:

- equidistant trace increment
- equal trace number for each 2D-line
- identical start coordinate for each 2D-line
- equidistant distance between the individual 2D-lines.

In the following the individual steps for creating such a 3D-file are described in detail:

1. enter the module **2D-dataanalysis**
2. activate the option **file/import**
3. choose the following options within the **import** menu:

data type: const. offset (version 2) or 3D-const.offset (from version 3)

ProfileDirection: direction of the original 2D-lines (enter either x or y)

ProfileConstant: constant coordinate direction of the original 2D-lines (enter either y or x depending on the chosen profile direction)

xstart: start coordinate in x-direction

ystart: start coordinate in y-direction

xend, yend, zstart, zend: not necessary to input

outputformat: 16 bit integer (e.g. for GPR-data) or 32 bit floating point for a higher data resolution (e.g. for seismic data).

To be considered for SEG Y 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.

filename specification: original name

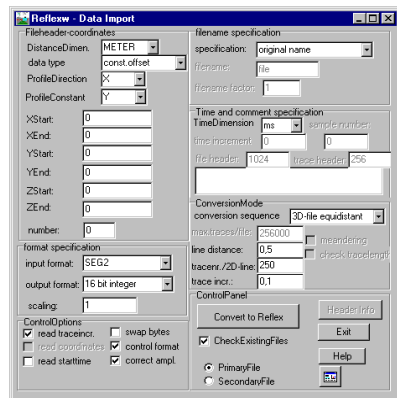
TimeDimension: ns (GPR data) or ms (seismic data)

conversion sequence: 3D-file equidistant

linedistance: enter the distance between the individual parallel 2D-lines

tracenr./2D-line: enter the number of traces for each 2D-lines containing within the 3D-file

traceincr.: enter the increment between the traces within each 2D-line



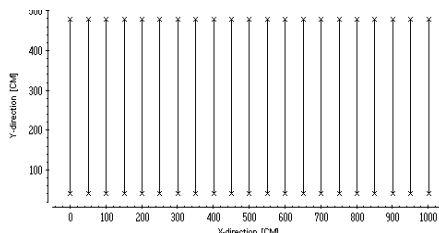
4. activate the option **Convert To Reflex**: the wanted original 3D-file must be chosen. The data within the resulting REFLEXW 3D-datafile are stored in the same manner like the original data but the ensemble number which controls the different 2D-lines is set based on the entered tracenr./2D-line.

0.9.1.1.2 Generating a 3D-file from Reflexw formatted parallel 2D-lines

The preconditions for generating a 3D-file within the 3D-datainterpretation without interpolation are REFLEX formatted 2D-lines with:

- parallel 2D-lines with equidistant profile increment
- equidistant trace increment
- equidistant time increment
- same number of points per trace
- same number of traces per 2D-line

If these preconditions are satisfied you may easily construct a 3D-file without a spatial interpolation:



1. enter the module **3D-datainterpretation**

2. activate the option **file/Generate 3D-file from 2D-files**

3. choose the following options within the **Generate 3D-file from 2D-lines** menu:

3D-filename: enter any name for the 3D-datafile

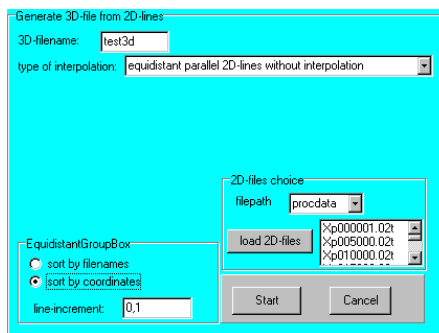
type of interpolation: equidistant parallel 2D-lines without interpolation

EquidistantGroupBox:

choose either sort by filenames or sort by coordinates (see item 4)

line-increment: enter the increment between the different 2D-lines

4. choose the wanted **filepath** and activate the option **load 2D-files** in order to choose the wanted 2D-files from the openfile dialog (multiple choice using the shift or str-key). The sorting of the 2D-files depends on the chosen option within the EquidistantGroupBox. With sort by filenames activated all chosen original datafiles are sorted with ascending filename order. To be considered: problems may occur if the files have names like file1, file2,..., file10, file11. In this case the choosen files are resorted in the following manner: file1, file10, file11, file12, ..., file2, ... which leads to a wrong sorting within the resulting 3D-file. With sort by coordinates activated all chosen original datafiles are sorted with ascending profileconstant coordinates. In this case the profileconstant coordinates must have been defined first within the fileheader.



5. Activating the option **start** starts the generation of the 3D-datafile. The 3D-file is generated without any interpolation. With ProfileDirection set to X and ProfileConstant set to Y the number of points in x-direction is equal the number of traces per 2D-file, the number of points in y-direction is equal the number of 2D-files and the number of points in z-(time-)direction is equal the number of points per trace. The file will be stored under the path rohdata under the current project directory. After having created the file the 3D-data are automatically loaded into the RAM (see also chap. 0.9.3). The 3D-datafile can be processed within the 2D-dataanalysis (see chap. 0.9.2).

0.9.1.2 Generating a 3D-file with interpolation (done within the 3D-datainterpretation)

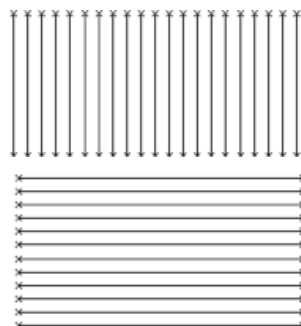
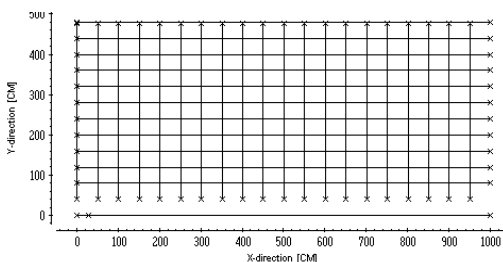
A 3D-datafile may also be generated from REFLEXW formatted 2D-lines. The 2D-lines may be raw data (then the processing is done for the 3D-datafile) or already processed data. There are no general rules whether the processing shall be done for the 3D-datafile or for the individual 2D-lines. If only few 2D-lines are present it might be better to do the processing for the 2D-lines.

REFLEXW allows to construct a 3D-datafile either from equidistant parallel or crossing 2D-lines (chap. 0.9.1.2.1) or from 2D-lines freely orientated within one acquisition plane (chap. 0.9.1.2.1) both using a weighted interpolation. In any case the coordinates (start- and endcoordinate in profiledirection and profileconstant coordinate) must have been defined within the file- or traceheader(s) of each 2D-line.

0.9.1.2.1 Generating a 3D-file from parallel or crossing 2D-lines

If the original data are not parallel equidistant lines of constant lengths but they have for example different lengths, different start coordinates or crossing lines you may use the interpolation routines within REFLEXW for generating a new 3D-datafile. You may also use this interpolation for parallel equidistant lines (see also chap. 0.9.1.1.2) if an interpolation is wanted (e.g. if only few 2D-lines exist).

To be considered: The necessary processing (see chap. 0.9.2.) should always be done onto the original 2D-datafiles. Therefore the 3D-datafile should be generated based on the completely processed 2D-datafiles.



generation flow:

1. enter the module **3D-datainterpretation**
2. activate the option **file/Generate 3D-file from 2D-files**
3. choose the following options within the **Generate 3D-file from 2D-lines** menu:

3D-filename: enter any name for the 3D-datafile

type of interpolation: use independent interpolation scheme for pure x- and y-2D-lines

3D coordinates group box :
XStart, XEnd, YStart and YEnd: specify the range coordinates within the dataacquisition plane for the computation of the 3D-data cube.

XRasterincrement and YRasterincrement: specify the grid interval of the two coordinate axes spanning the plane in the given distance dimension. A

useful value for both XRasterincrement and YRasterincrement is the traceincrement used for the individual 2D-profiles.

XInterpolation and YInterpolation: define the interpolation area (a rectangle) in the given distance dimension. To guarantee a complete filing of the 3D-data cube the interpolation range in x- and y-direction has to comply with the greatest occurring distance between two data points in the corresponding direction whereby a too large range leads to an averaging.

interpol.weight: allows to specify the type of the interpolation weight.

timerange/sorting group box :

time end: specifies the timerange for the 3D-data. The start time is always 0.

sorting: determines the sorting of the profiles entering the computation of the time slices. Four different sortings are possible: fileheader coordinates or receiver coordinates, midpoint coordinates and CMP coordinates (defined within the individual traceheaders).

4. choose the wanted **filepath** and activate the option **load 2D-files** in order to choose the wanted 2D-files from the openfile dialog (multiple choice using the shift or str-key).

5. Activating the option **start** starts the generation of the 3D-datafile. The number of datapoints depends on the entered range of the 3D-cube and the raster increments. To ensure, that the resulting 3D-file does not exceed the max. size of 2048^3 points, the parameters XRasterincrement and YRasterincrement can be enlarged and/or the volume of the data to be considered can be reduced. The file will be stored under the path `rohdata` under the current project directory. After having created the file the 3D-data are automatically loaded into the RAM (see also chap. 0.9.3). The 3D-datafile can be processed within the 2D-dataanalysis (see chap. 0.9.2).

Generate 3D-file from 2D-lines

3D-filename: test3d

type of interpolation: use independent interpolation scheme for pure x- and y-2D-lir

☐ add x-y-scans by choosing the max. abs. value

☒ apply mean filter in time direction 1

3D coordinates

XStart: 0

XEnd: 0,8000000115

YStart: 0

YEnd: 0,8000000115

XRasterincrement: 0,005

YRasterincrement: 0,005

XInterpolation: 0,15

YInterpolation: 0,15

interpol.weight: square weight

data scaling: 1

timerange/sorting

time end: 7

timeincrement scale: 1

sorting: receiver coordinates

2D-files choice

filepath: rawdata

filerfilter: *.dat

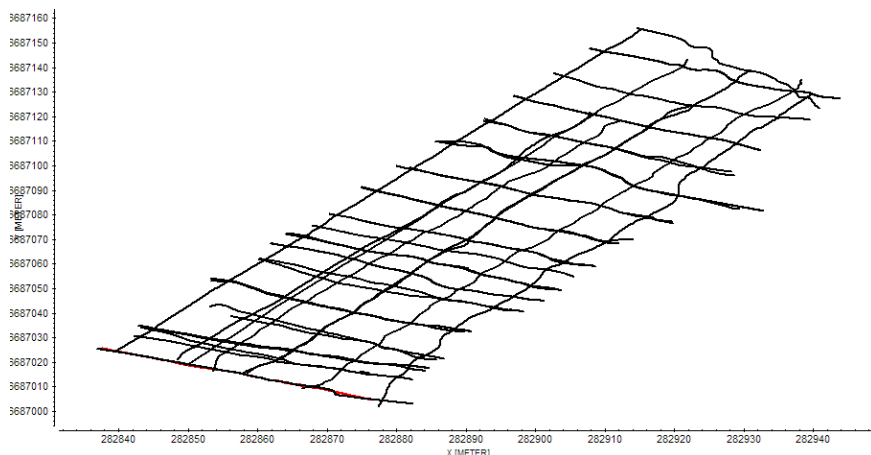
load 2D-files: X01.DAT, X02.DAT, X03.DAT

min/max.xy coord: X01.DAT, X02.DAT, X03.DAT

☒ update traceheader coordinates

Start Cancel

0.9.1.2.2 Generating a 3D-file for freely distributed 2D-lines



If the original data are freely distributed within the acquisition plane you must use the interpolation routines within REFLEXW for generating a new 3D-datafile. To be considered: The necessary processing (see chap. 0.9.2.) should always be done onto the original 2D-datafiles. Therefore the 3D-datafile should be generated based on the completely processed 2D-datafiles.

generation flow:

1. enter the module **3D-datainterpretation**

2. activate the option **file/Generate 3D-file from 2D-files**

3. choose the following options within the **Generate 3D-file from 2D-lines** menu:

3D-filename: enter any name for the 3D-datafile

type of interpolation: use interpolation scheme for freely distributed 2D-lines

3D coordinates group box :
XStart, XEnd, YStart and YEnd: specify the range coordinates within the

Generate 3D-file from 2D-lines

3D-filename:

type of interpolation:

☐ add x-y-scans by choosing the max. abs. value

☒ apply mean filter in time direction:

3D coordinates

XStart:

XEnd:

YStart:

YEnd:

XRasterincrement:

YRasterincrement:

XInterpolation:

YInterpolation:

interpol. weight:

data scaling:

timerange/sorting

time end:

timeincrement scale:

sorting:

2D-files choice

filepath:

filefilter:

load 2D-files:

min/max.xy coord:

☒ update traceheader coordinates

dataacquisition plane for the computation of the 3D-data cube.

XRasterincrement and YRasterincrement: specify the grid interval of the two coordinate axes spanning the plane in the given distance dimension. A useful value for both XRasterincrement and YRasterincrement is the traceincrement used for the individual 2D-profiles.

XInterpolation and YInterpolation: define the interpolation area (a rectangle) in the given distance dimension. To guarantee a complete filing of the 3D-data cube the interpolation range in x- and y-direction has to comply with the greatest occurring distance between two data points in the corresponding direction whereby a too large range leads to an averaging.

interpol.weight: allows to specify the type of the interpolation weight.

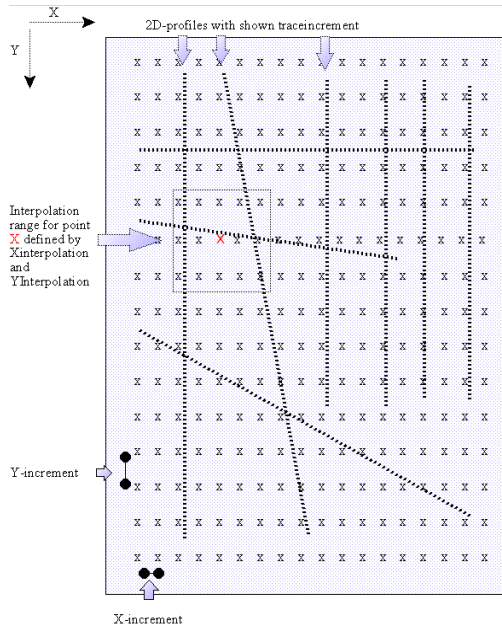
timerange/sorting group box :

time end: specifies the timerange for the 3D-data cube. The time always starts at 0.

sorting: determines the sorting of the profiles entering the computation of the time slices.

Four different sortings are possible: fileheader coordinates or receiver coordinates, midpoint

coordinates and CMP coordinates (defined within the individual traceheaders).

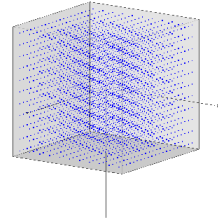


4. choose the wanted **filepath** and activate the option **load 2D-files** in order to choose the wanted 2D-files from the openfile dialog (multiple choice using the shift or str-key).

5. Activating the option **start** starts the generation of the 3D-datafile. The number of datapoints depends on the entered range of the 3D-cube and the raster increments. To ensure, that the resulting 3D-file does not exceed the max. size of 2048^3 points, the parameters XRasterincrement and YRasterincrement can be enlarged and/or the volume of the data to be considered can be reduced. The file will be stored under the path rohdata under the current project directory. After having created the file the 3D-data are automatically loaded into the RAM (see also chap. 0.9.3). The 3D-datafile can be processed within the 2D-dataanalysis (see chap. 0.9.2).

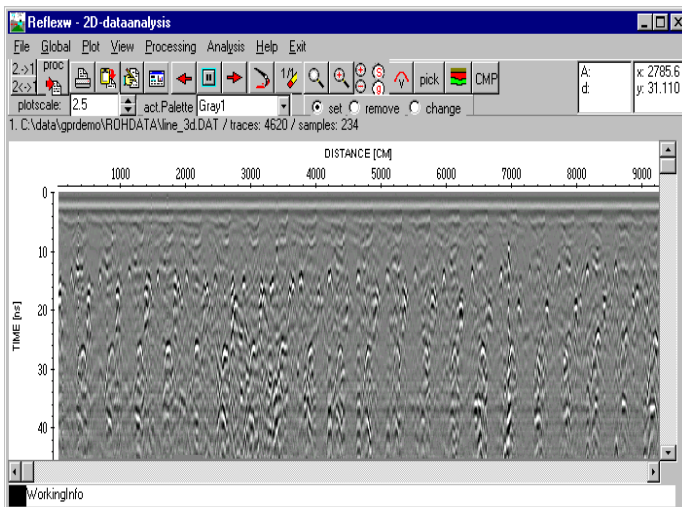
0.9.1.3 resulting REFLEXW 3D-datafile

The resulting 3D-file is a new file with equidistant rasterincrements in x, y or z (time) direction. The 3 rasterincrements may differ. The max. number of points in one direction is 2048, the max. size is 2048*2048*2048 points. The resulting 3D-file has the same REFLEXW format like a 2D-datafile and may be processed within the 2D-dataanalysis module (chap. 0.9.2.) whereby the interpretation must be done within the 3D-datainterpretation (chap. 0.9.3).



The different 2D-lines are stored sequentially. The distance axis within the 2D-dataanalysis represents the distance sum over all 2D-lines. Because of the normally huge number of traces and the chosen plotmode (Pointmode, XYScaledPlot) the screen display resolution may be too small to plot the data correctly (XScale too small). In this case choose the Zoom-option in order to only display a small part of the data in x-direction.

To be considered: If only few 2D-lines have been acquired with a large number of traces the resulting 3D-datafile may have many points in one direction and very few in the other. In this case it might be better to use the interpolation method for generating a 3D-datafile from REFLEXW formatted 2D-lines (see chap. 0.9.1.2.1) especially if you want to display timeslices. Although it is clear that the resolution will not be enhanced the timeslices may appear more reasonable by the averaging of the interpolation.



0.9.1.4 combine different 3D-datafiles

The option generate 3D-file from 2D-lines as described in chap. 0.9.1.2.2 can also be used in order to combine different 3D-datafiles into one 3D-datafile. It is recommended that all 3D-datafiles have the same trace- and lineincrements but in principle they can also differ. The calculation must be based on the traceheader xy-receiver coordinates. Therefore the following processing flow must be done:

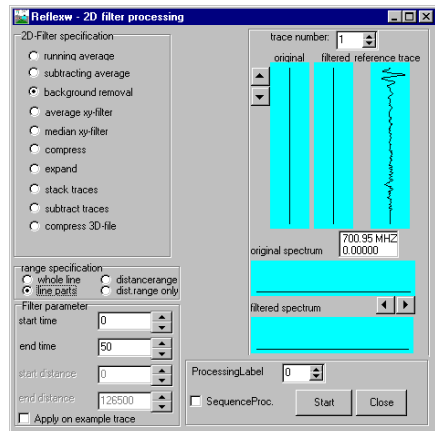
1. Load the first 3D-file within the 2D-dataanalysis and enter the edit **fileheader** menu and define the correct start x and y-coordinate if not already done. Set the data type to **3D const. offset** and click on save.
2. Click on show traceheader and update the traceheader coordinates for using the update option **fileheader-3D** within the trace header menu.
3. Do step 1 and 2 for all wanted 3D-files.
4. Enter the **3D-datainterpretation** module and choose the option **generate 3D-file from 2D-lines** and activate the interpolation type use interpolation scheme for freely distributed 2D-lines. Enter the overall xy-range and the x- and y-raster increments. Useful values for the x- and y-increments are the traceincrement (corresponds to x-increment for profiledirection X and y-increment for profiledirection Y) and the lineincrement (corresponds to x-increment for profileconstant X and y-increment for profileconstant Y) of the original 3D-datafiles. If all 3D-files have the same increments the Xinterpolation and Yinterpolation can be set to the same value of the rasterincrements. In this case no interpolation will be done. Enter the time end and set sorting to receiver coordinates. Choose the wanted 3D-files using the option load 2D-files and enter a 3D-filename and click on start. The resulting 3D-file contains all informations of the chosen 3D-files within the given overall xy-range.

0.9.2 processing a 3D-data file (done within the 2D-dataanalysis)

A **3D-data file** is a single REFLEX formatted file which consists of several equidistant 2D-cuts sequentially stored. All traces belonging to one 2D-cut have the same ensemble-number which is stored within the REFLEXW traceheader. The direction (x or y) within the fileheader determines the direction of the 2D-cuts.

A 3D-data file may be processed like any 2D-file (see for example filtering guide) but there are some peculiarities to be considered for:

1. If the 3D-datafile is generated by an interpolation (chap. 0.9.1.2) the complete processing should be always applied onto the original 2D-datafiles.
2. If the data contain hyperbola from small scale objects or from pipes a migration is necessary in addition if timeslices are used for the interpretation (see also chap. 0.9.3.).
3. The processing step static correction is very important because the same time baseline must be used for all 2D-cuts. Otherwise the timeslices are not correct.
4. multichannel filters like background removal or fk-filter:
the option **line parts** should be activated. In this case the distance range for applying the filter is automatically determined for line parts (individual 2D-cuts) separated by each other by a new ensemble number.



0.9.3 interpretation of a 3D-data file (done within the 3D-datainterpretation)

The module 3D data-interpretation allows the interpretation of 3-dimensional data by displaying x-, y- or z-slices or the full 3D-data volume.

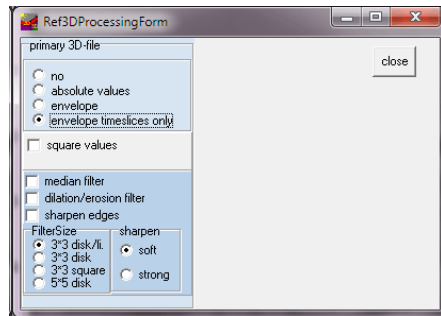
The data are completely loaded into the RAM of the computer whereby a fast visualization of the data is possible. The max. number of points in each direction (x-cuts, y-cuts and z-(time)cuts) is 2048. Three different display options are available (option windows, option scroll and option 3D-cube - see below).

0.9.3.1 load a 3D-datafile

1. enter the **3D-datainterpretation**

2. load the 3D-datafile using the option **File/Open 3D-file**

3. After having chosen the wanted 3D-file the Ref3DProcessingForm menu opens which allows you to define the internal processing of the 3D-file. Use **envelope** if you want to display timeslices. Use **no** if you want to interpret x- or y-cuts.



4. activate the option **close** and the chosen 3D-file is loaded into the RAM.

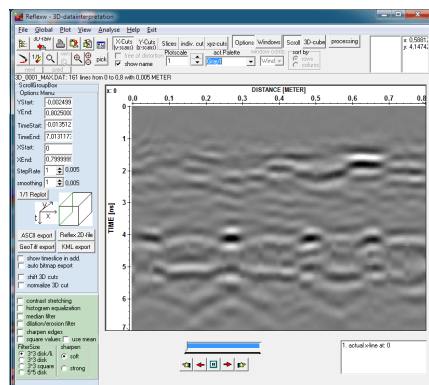
5. The internal processing may be changed at a later stage using the option **processing**.

0.9.3.2 display a 3D-datafile using the scroll option

1. after having loaded a 3D-datafile activate the option **scroll**

2. Activating one of the three different **cuts** (X-Cuts, Y-Cuts or Slices) allows to scroll through the 3D-databcube into the given direction using the scroll bar or the arrows. The option xyz-cuts allow the simultaneous display of the 3 cuts (see chap. 0.9.3.2.1)

3. The coordinate parameters within the Options Menu are automatically set. The option StepRate allows you to change step rate for scrolling. Every step range



2D-cut will be plotted when scrolling.

The option smoothing allows to enter a value for smoothing the data in the scroll direction. A value of 1 means no smoothing. A value of 2 means that 2D-cuts are summed up and the mean values are calculated. The smoothing parameter is independent from the StepRate parameter. The smoothing is done symmetrically in both directions (half the smoothing value in each direction).

0.9.3.2.1. xyz-cuts option

1. The option **xyz-cuts** allows different simultaneous display possibilities of the 3 cuts. The actual positions of the other cuts are indicated by a large cross within each cut.

A fast interactive change of the cuts is done by several mouse actions:

- the mouse wheel can be used in order to go through any of the cuts
- pressing the left mouse button allows to control the other cuts within any cut.

2. The **grouping** box allows the following display types:

- **by geometry** (see picture below). The z-cut (slice) is displayed on the top/left surrounded by the x- and y-cut using a free of distortion scale and using a time axis scale which automatically takes into account the velocity entered for the depth axis within the plotoptions menu or within the xyz-cut option panel.

- **by columns/rows** (see picture above). The xy-cuts are displayed right to the z-cut (slice). The time axis scale of the xy-cuts may be adjusted to the xy-scale activating the option **time axis free of distortion**

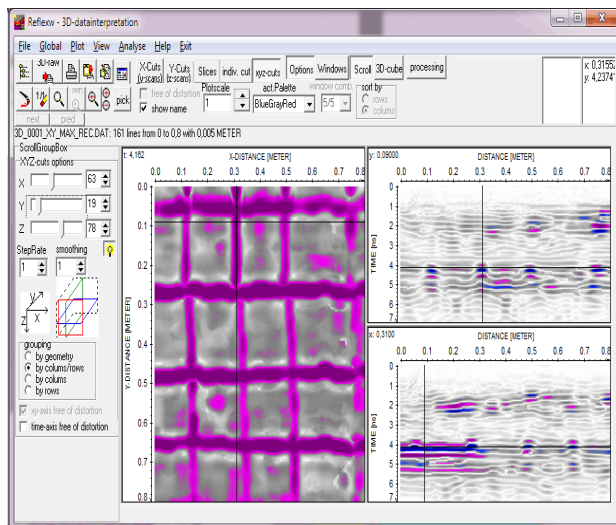
- **by rows/columns**.

The xy-cuts are displayed below the z-cut (slice). The time axis scale of the xy-cuts may be adjusted to the xy-scale activating the option **time axis free of distortion**

- **by columns** (see picture below): all three cuts will be displayed within one row with the z-cut (slice) in the middle

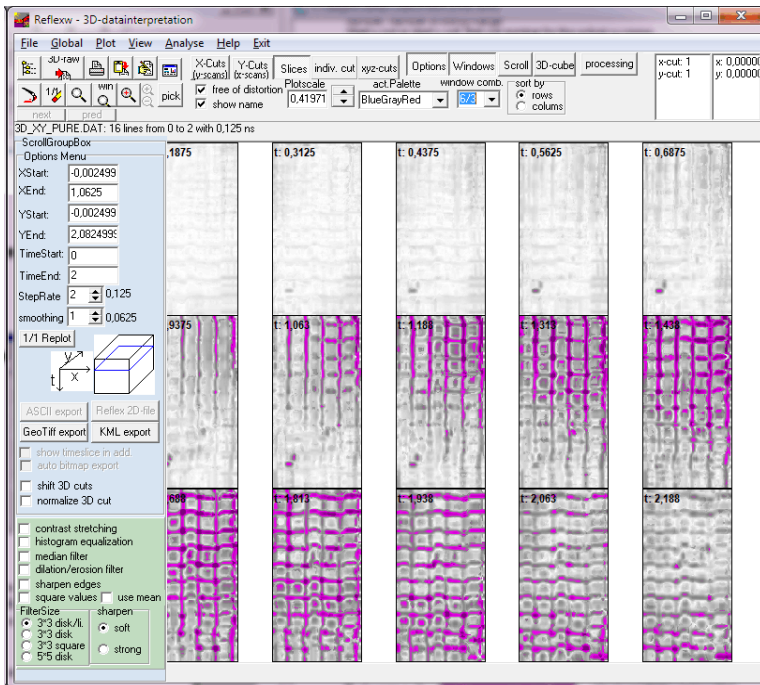
- **by rows** (see picture below): all three cuts will be displayed within one column with the z-cut (slice) in the middle

Activating one of the three different **cuts** (X-Cuts, Y-Cuts or Slices) allows to scroll through the 3D-databcube into the given direction using the scroll bar or the arrows.



0.9.3.3 display a 3D-datafile using the windows option

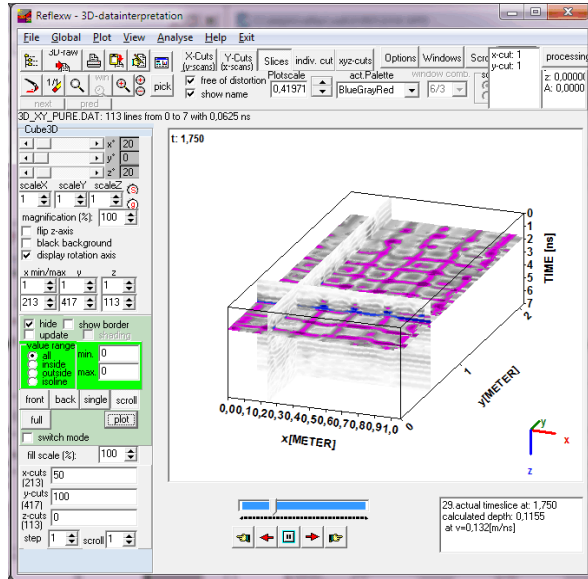
1. after having loaded a 3D-datafile activate the option **windows**
2. Activating one of the three different **cuts** (X-Cuts, Y-Cuts or Slices) allows to display several 2D-cuts of the 3D-cube, placed in freely choosable manner within the working window.
3. The **coordinate** parameters within the Options Menu are automatically set. The max. number of cuts to be displayed simultaneously is 25. The program automatically calculates the StepRate for the given cut range (in this example TimeStart and TimeEnd) and calculates the increment (this example 0.125 ns). You may manually change these parameters.
4. The option **window comb.** allows to choose the combination of the windows to be subdivided in horizontal and vertical direction. the possible combinations are automatically predefined. If you have 24 different cuts to be displayed you may choose the combination 8/3 - this means the screen is subdivided into 8 horizontal and 3 vertical cuts.
5. The option **Options** can be deactivated. In this case the OptionsMenu disappears and the full screen is visible.



0.9.3.4 display a 3D-datafile using the 3D-cube option

1. after having loaded a 3D-datafile activate the option **3D-cube**
2. The 3D-cube will be displayed together with the Cube3DOptions menu.

3. The **size** of the 3D-cube is determined from the number of the points into the different directions **x**, **y** and **z**(time). The point numbers are displayed within the options min/max for **x**, **y** and **z**. Here you also may restrict the range. The axis scales of the 3D-cube are based on these values and do not correspond to the real coordinates. This may significantly differ for example if the 3D-datafile has been created without interpolation (traceincrement and lineincrement differ significantly). It is possible to change the axis scales using the options **ScaleX**, **ScaleY** and **ScaleZ**.



4. You have the possibility to look at the cube from different angles, e.g. from the front side, the back side or from the side. The angles can be changed manually within the options **x°**, **y°** and **z°** or using the mouse with pressed left mouse button within the 3D-cube display.

5. You have different display possibilities. You may select if only the front or back planes of the data cube are displayed (options **front** and **back**) or the full 3D-data volume (option **full**). In addition you only may select single cuts (option **single**) and scroll (option **scroll**) through the cube in one distinct direction.

0.9.4 3D-view of single 2D-lines with arbitrary geometry

In many cases only few 2D-lines with different geometry (parallel or crossing or any other angle) have been acquired which you also want to display within a 3D-cube. The following chapter describes how to manage such a 3D-view of single 2D-lines with arbitrary geometry. Precondition is that the geometries of the 2D-lines have been specified either within the fileheaders or within the individual traceheaders.

1. enter the module **3D-datainterpretation**
2. activate the option **file/Generate 3D-file from 2D-files**
3. choose the following options within the **Generate 3D-file from 2D-lines** menu:

3D-filename: enter any name for the 3D-datafile

type of interpolation: use interpolation scheme for freely distributed 2D-lines

3D coordinates group box :

XStart, XEnd, YStart and YEnd:

specify the range coordinates within the dataacquisition plane for the computation of the 3D-data cube.

XRasterincrement and

YRasterincrement: specify the grid interval of the two coordinate axes

spanning the plane in the given distance dimension. A useful value for both XRasterincrement and YRasterincrement is the traceincrement used for the individual 2D-profiles. The rasterincrements should not be smaller than the increments within the original 2D-lines.

XInterpolation and YInterpolation: enter the same values as for XRasterincrement and YRasterincrement in order to avoid any interpolation.

interpol.weight: allows to specify the type of the interpolation weight. It has no meaning if the interpolation values match the rasterincrements.

timerange/sorting group box :

time end: specifies the timerange for the 3D-data cube. The time always starts at 0.

sorting: determines the sorting of the profiles entering the computation of the time slices. Four different sortings are possible: fileheader coordinates or receiver coordinates, midpoint coordinates and CMP coordinates (defined within the individual traceheaders).

4. choose the wanted **filepath** and activate the option **load 2D-files** in order to choose the wanted 2D-files from the openfile dialog (multiple choice using the shift or str-key).

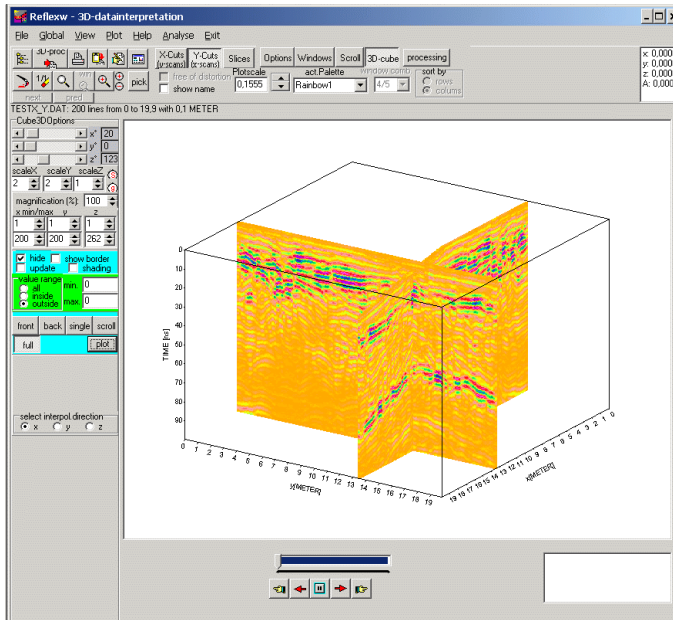
5. Activating the option **start** starts the generation of the 3D-datafile. The number of datapoints depends on the entered range of the 3D-cube and the raster

increments. To ensure, that the resulting 3D-file does not exceed the max. size of 2048^3 points, the parameters `XRasterincrement` and `YRasterincrement` can be enlarged and/or the volume of the data to be considered can be reduced. The file will be stored under the path `rohdata` under the current project directory. After having created the file the 3D-data are automatically loaded into the RAM (see also chap. 0.9.3). The resulting 3D-file contains 0 amplitude values at all positions where no original data are present.

6. Activate the option **3D-cube**. The other 2 display modes (scroll and windows) may not be very useful because of the 0 values at the non originally covered positions. Use any of the available scaling and rotating options in order to get the best 3D-view.

7. Activate the option **outside** within the `Cube3DOptions` box and enter the **value range** 0 for **min.** and 0 for **max.** This ensures that all 0 values will not be taken into account when plotting the 3D-data.

8. Activate the option **full** and plot the data by pressing the option **plot**.



9. If any other combination of the original 2D-lines is wanted a new 3D-file must be generated (items 3-8).

0.10 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. 0.10.2)

2D ray tracing (chap. 0.10.3)

2D- and 3D- transmission and refraction tomography (chap. 0.10.4)

Chap. 0.10.1 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 guide the instructions within the corresponding chapters of this manual and the online help.

0.10.1 model generation

0.10.1.1 create a new model

First the generation of a complete new model is described.

1. Enter the module **modelling**

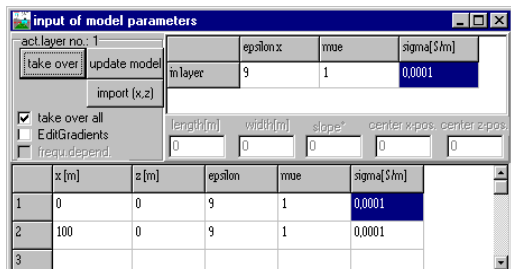
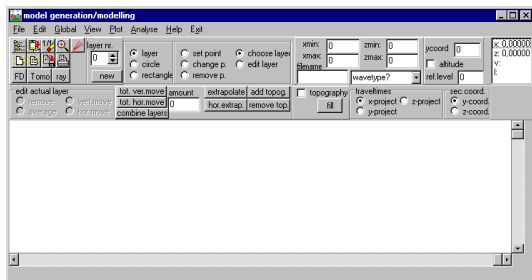
2. Choose the **wavetype** (e.g. electromagnetic for a GPR-modelling or acoustic for a raytracing).

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 current 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 instructions under the individual chapters of this manual or to the online help.

0.10.1.2 change an existing model

In this chapter the changing of an existing model will be described. Such a model may be built up as described in the previous chapter or it may result from a seismic refraction inversion (see chap. 0.7.2).

There are different possibilities to change the existing model. A few of them are described below:

First it might be useful to restrict the number of points of each layer. This will be done using the option **edit layer**. Activate this option and then activate the option **average**. Enter an average value (e.g. 5 for amount) and click on the wanted layer boundary (left mouse button). Then this layer boundary will only consist of a 5 times fewer datapoints. The option may be applied for each layer boundary. After this option it might be useful to extrapolate the boundary either using the option **hor.extrap.** or **extraplate**.

Activating the option **vert.move** allows you to move up or down a special layer boundary (all depth values by the same amount).

In order to change the individual datapoints of a distinct layer activate the option **choose layer** and click on the wanted layer - the layer will be highlighted (in red). Now you may either change the layer boundary or the velocity within the layer.

When clicking on the right mouse the **input parameter window** appears and you may change the velocities for the individual datapoints. You may enter the new values either within the table or within the upper right overall table which allows you to change the velocities for all points (option "take over all" activated) when clicking on the option **take over**.

In order to change the individual depths of the layer boundary activate the option **change p.** and click on the wanted boundary datapoint and then move this datapoint with pressed mouse to the wanted position.

The option **remove p.** allows to remove any layerboundary points, **set p.** includes a new layerboundary point.

0.10.2 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).

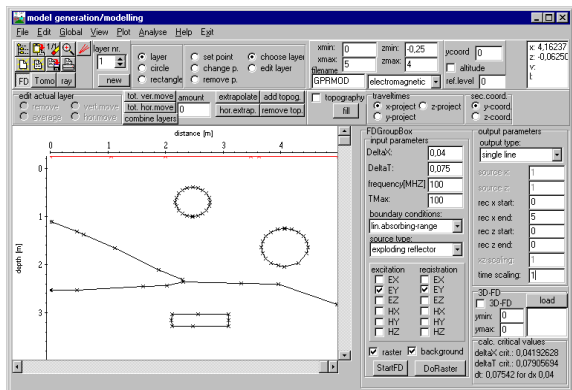
1. First a new model must be generated (see chap. 0.10.1) or an already existing model must be loaded using the option file/load model.

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: $\Delta t \leq 1./(\sqrt{2}V)$ for the el.magnetic propagation and $\Delta t \leq 1./(Vp+Vs)$ 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 background 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.

0.10.3 ray tracing

The ray tracing modelling tool allows the traveltimes 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. 0.10.2).

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

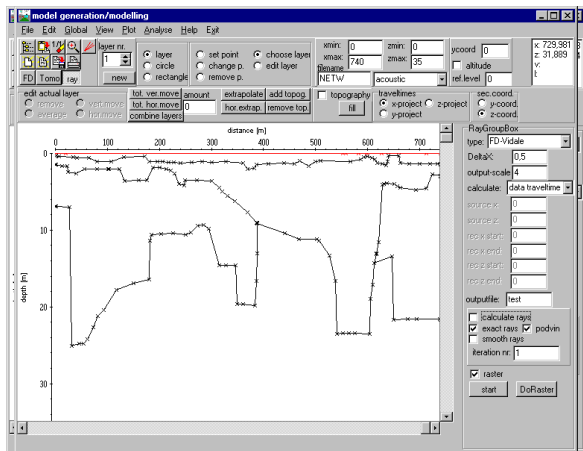
In the following the application to seismic refraction data is described.

1. first a new model must be generated (see chap. 0.10.1) 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.



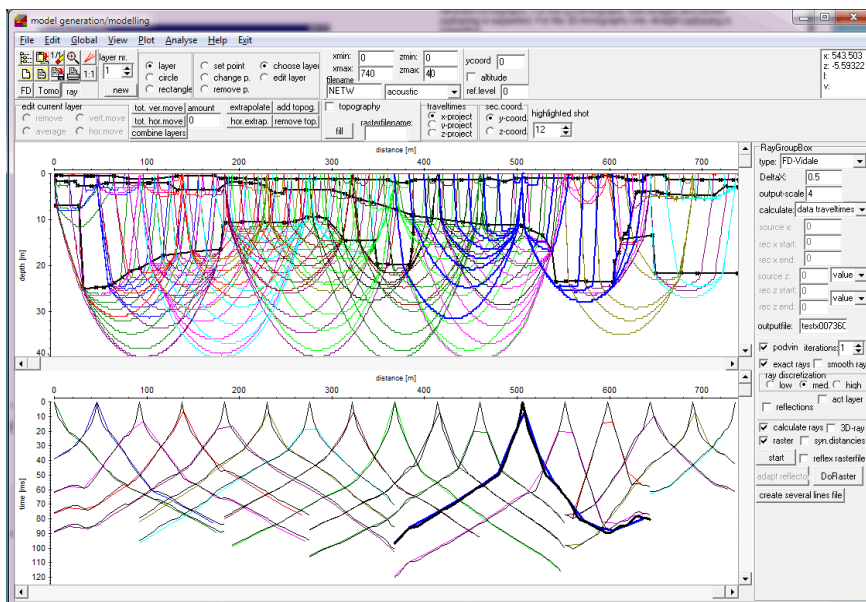
5. Now the ray tracing parameters have to be chosen:

- 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. Choose a shot number for highlighting using the option **highlighted shot**. You can interactively go through the highlighted shot numbers in order to analyse the differences of the individual shots.

You may also check for the mean traveltime difference using the option **Analyse/calculate traveltime differences using actual coord.projection**.



8. 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.

0.10.4. 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 0.10.4.1 includes the format and the picking of the **traveltime data**. In chapter 0.10.4.2 the 2D tomographic interpretation of **borehole-borehole transmission tomography** is described. In chapter 0.10.4.3 the 2D tomographic interpretation of a **refraction tomography** is described.

0.10.4.1 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.

0.10.4.1.1 single shot analysis

Depending 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

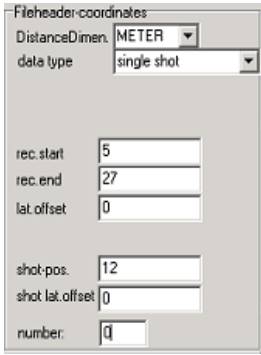

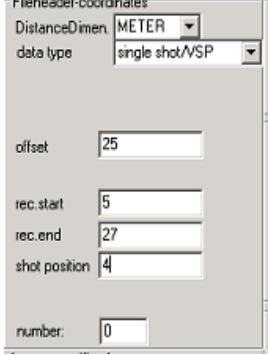
single shot/boreholes: borehole/borehole geometry - shots and receivers are located within two parallel boreholes

single shot/VSP: borehole/surface geometry - the receivers are located within a borehole, the shots are located at the surface

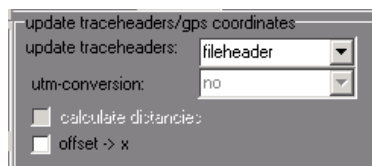
The shotfiles can be imported and handled completely separately or using a conversion sequence.

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
<p>The equidistant receivers are located at the surface from 5 to 27 m. The shot is placed at 22 m. The lateral offsets of the receivers and the shot are 0.</p>	<p>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.</p>	<p>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.</p>
		

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



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 possibilities 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-nt.	distance	shot-x	shot-y	shot-z	rec.-x	rec.-y	rec.-z	time delay	gain	time collect
1	10	3	15	0	31	10	0	0	1	0
2	11	3	15	0	31	11	0	0	1	0
3	12	3	15	0	31	12	0	0	1	0
4	13	3	15	0	31	13	0	0	1	0
5	14	3	15	0	31	14	0	0	1	0
6	15	3	15	0	31	15	0	0	1	0
7	16	3	15	0	31	16	0	0	1	0
8	17	3	15	0	31	17	0	0	1	0
9	18	3	15	0	31	18	0	0	1	0
10	19	3	15	0	31	19	0	0	1	0
11	20	3	15	0	31	20	0	0	1	0
12	21	3	15	0	31	21	0	0	1	0

TopographyGroupBox

topography (x,z values)

☒ update shot z-pos.

☒ update receiver z-pos.

☒ use x

☐ apply x-z topography

☐ get distance along topography

EditGroupBox

apply borehole deviations

3D-view of boreholes

smooth rec. xy-coord.

factor f.smooth: 4

source <-> rec.

x <-> y

project on x

interpolate

fileheader coordinates

data type: single shot/boreho

shot-pos: 15

shot-offset 3

rec.start 10

rec.end 21

rec.offset 31

UpdateGroupBox

load from AsciiFile

save on AsciiFile

update from fileheader

update fileheader

update distances

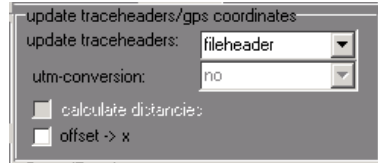
reload from file

save changes

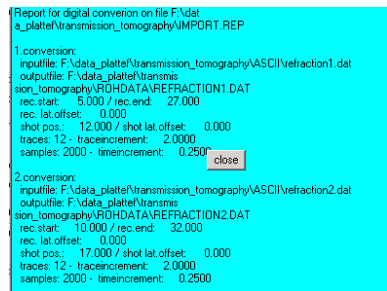
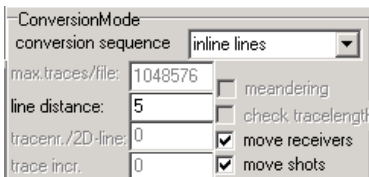
close

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 coordinates box. After having activated the option convert to Reflexw you must select all original files which fall in the selected geometry layout using the shift or ctrl key. In the case of the conversion sequence inline lines the edit traceheader coordinates menu does not open after having converted the files.

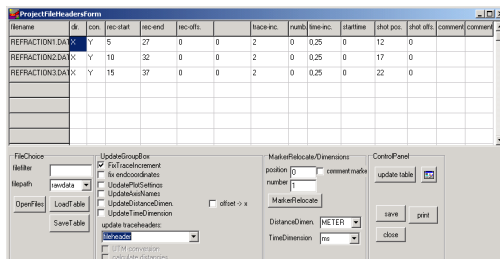


Surface/surface layout 1 with moving receivers and shots (e.g. seismic refraction or reflection data). For a fixed line with moving shots simply deactivate move receivers.



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.



Borehole/borehole layout 2 with a start position within the borehole for the shot at 15 m and moving shots upwards with an increment of 5 m:

After having imported all files it is possible

ConversionMode
 conversion sequence inline lines
 max.traces/file: 1048576
 line distance: -5
 tracer./2D-line: 0
 trace incr. 0
☐ meandering
☐ check tracelength
☐ move receivers
☒ move shots

Report for digital conversion on file F:\data
 a_plate\transmission_tomography\IMPORT.REP

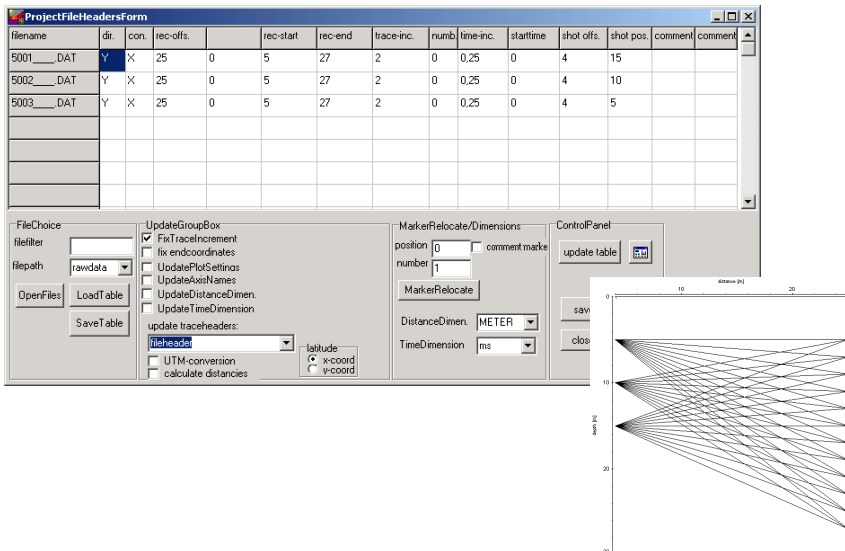
1.conversion:
 inputfile: F:\data_plate\transmission_tomography\ASCII\5001.dat
 outputfile: F:\data_plate\transmission_tomography\AR0HDATA\5001____.DAT
 rec. offset: 25.000
 rec.start: 5.000 / rec.end: 27.000
 shot offset: 4.000 / shot position: 15.000
 traces: 12 - traceincrement: 2.0000
 samples: 2000 - timeincrement: 0.2500

2.conversion:
 inputfile: F:\data_plate\transmission_tomography\ASCII\5002.dat
 outputfile: F:\data_plate\transmission_tomography\AR0HDATA\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_plate\transmission_tomography\ASCII\5003.dat

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.



Borehole/surface layout 3 with a start position within 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 tracelengt

☐ move receivers

☒ move shots

Report for digital conversion on file F:\data_a_platte\transmission_tomography\IMPORT.REP

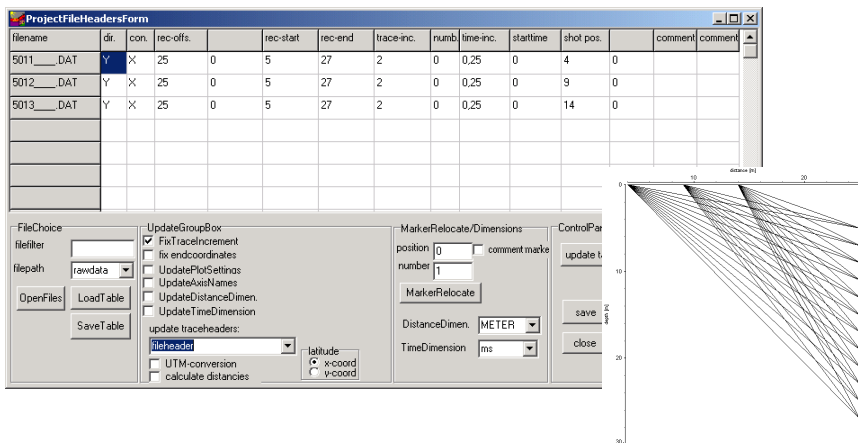
1.conversion:
inputfile: F:\data_platte\transmission_tomography\ASDI\5011.dat
outputfile: F:\data_platte\transmission_tomography\ROHDATA\5011____.DAT
rec. offset: 25.000
rec. start: 5.000 / rec.end: 27.000
shot offset: 4.000
traces: 12 - traceincrement: 2.0000
samples: 2000 - timeincrement: 0.2500

2.conversion:
inputfile: F:\data_platte\transmission_tomography\ASDI\5012.dat
outputfile: F:\data_platte\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_platte\transmission_tomography\ASDI\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.

0.10.4.1.2 several shots analysis

Putting all shots together has several advantages:

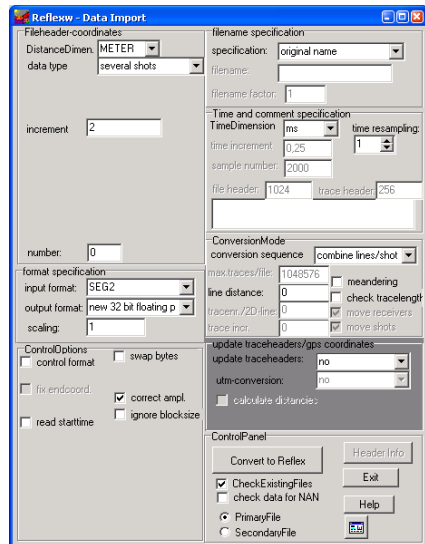
- setting the geometry is easier
- picking is faster

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:

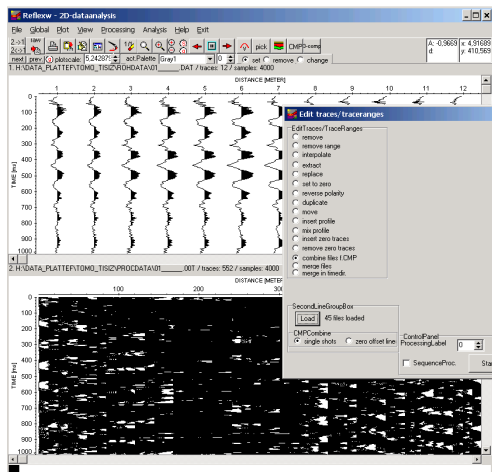
1. **Import** your single shot data. Set datatype to several shots because in step 2 all shots will be combined within one single file. The order of the files must be correct (e.g. file001.sg2 and not file1.sg2) because otherwise the sorting of the files is not correct for step number 2.

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 coordinates 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. Shot offset specifies the x-position of the shot borehole and receiver offset specifies the location of the receivers in y-direction (normally 0 for surface).

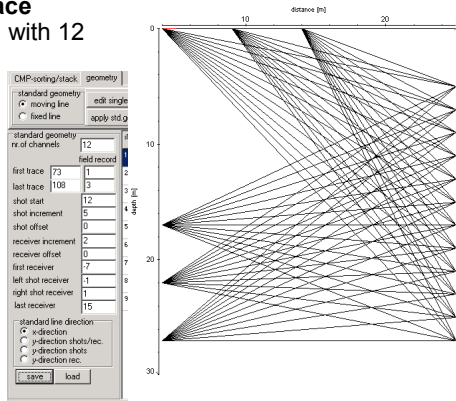
y-direction rec. activated: the receivers are assumed to be orientated in y-direction (borehole/surface layout). Use this option for example to define the geometry of a borehole containing the receivers and the shots placed at the surface. Receiver offset specifies the x-position of the receiver borehole 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.

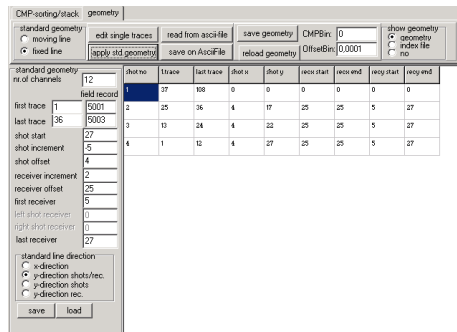
The parameters for the **surface/surface** configuration (layout 1 of chap. IV.1.1 with 12 receivers) are the following:

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



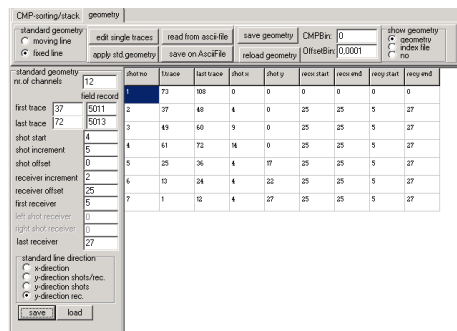
The parameters for the **borehole/borehole** configuration (layout 2 of chap. IV.1.1 with 12 receivers) are the following:

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



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.

75.6100	14.0000	0.0000	25.0000	21.0000	23.7065	313.54
75.6100	14.0000	0.0000	25.0000	23.0000	25.4951	337.19
75.6100	14.0000	0.0000	25.0000	25.0000	27.3130	361.24
75.6100	14.0000	0.0000	25.0000	27.0000	29.1548	385.59
75.6100	17.0000	0.0000	10.0000	0.0000	7.0000	92.58
75.6100	17.0000	0.0000	12.0000	0.0000	5.0000	66.13
75.6100	17.0000	0.0000	14.0000	0.0000	3.0000	39.68
75.6100	17.0000	0.0000	16.0000	0.0000	1.0000	13.23
75.6100	17.0000	0.0000	18.0000	0.0000	1.0000	13.23
74.3900	17.0000	0.0000	20.0000	0.0000	3.0000	40.33
74.3900	17.0000	0.0000	22.0000	0.0000	5.0000	67.21
74.3900	17.0000	0.0000	24.0000	0.0000	7.0000	94.10
74.3900	17.0000	0.0000	26.0000	0.0000	9.0000	120.98
74.3900	17.0000	0.0000	28.0000	0.0000	11.0000	147.87
74.3900	17.0000	0.0000	30.0000	0.0000	13.0000	174.75
74.3900	17.0000	0.0000	32.0000	0.0000	15.0000	201.64
74.3900	22.0000	0.0000	15.0000	0.0000	7.0000	94.10
74.3900	22.0000	0.0000	17.0000	0.0000	5.0000	67.21
74.3900	22.0000	0.0000	19.0000	0.0000	3.0000	40.33
73.1700	22.0000	0.0000	21.0000	0.0000	1.0000	13.67
73.1700	22.0000	0.0000	23.0000	0.0000	1.0000	13.67
73.1700	22.0000	0.0000	25.0000	0.0000	3.0000	41.00
73.1700	22.0000	0.0000	27.0000	0.0000	5.0000	68.33
71.9500	22.0000	0.0000	29.0000	0.0000	7.0000	97.29
70.7300	22.0000	0.0000	31.0000	0.0000	9.0000	127.24
70.7300	22.0000	0.0000	33.0000	0.0000	11.0000	155.52
71.9500	22.0000	0.0000	35.0000	0.0000	13.0000	180.68
71.9500	22.0000	0.0000	37.0000	0.0000	15.0000	208.48
82.9300	4.0000	17.0000	25.0000	5.0000	24.1868	291.65

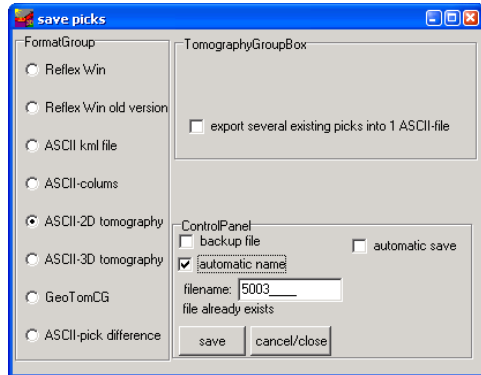
0.10.4.1.3 picking the first arrivals

After having imported the data and having defined the traceheader geometry the traveltimes 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.



0.10.4.2 performing the transmission tomography

1. First a starting model must be generated (see chap. 0.10.1) 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 chap. 0.10.4.1). If the 3D-data format is used for the 2D-tomography 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. The ASCII-tomography data may be created within the modules

- a. 2D-dataanalysis - save picks using the formats ASCII-2D tomography or ASCII-3D tomography
- b. traveltimes analysis - option export to ASCII
- c. modelling - export data traveltimes to ASCII

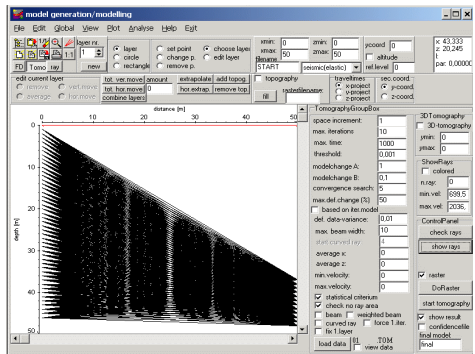
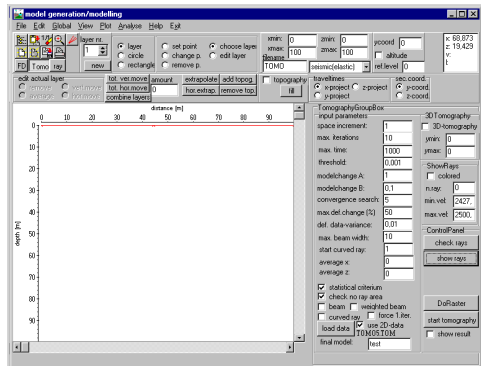
- Check the geometry of your loaded **traveltimedata** using the option **show rays**.

- 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.

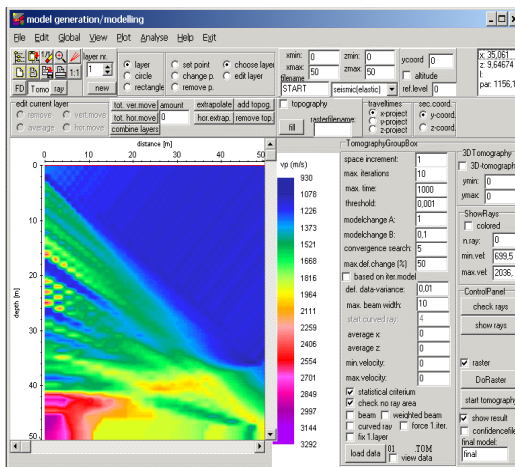
- For a first tomographic result you may use the other default parameters. There are no general rules for these parameters but you have to adapt the parameters to your data in order to get the best result.

- 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.

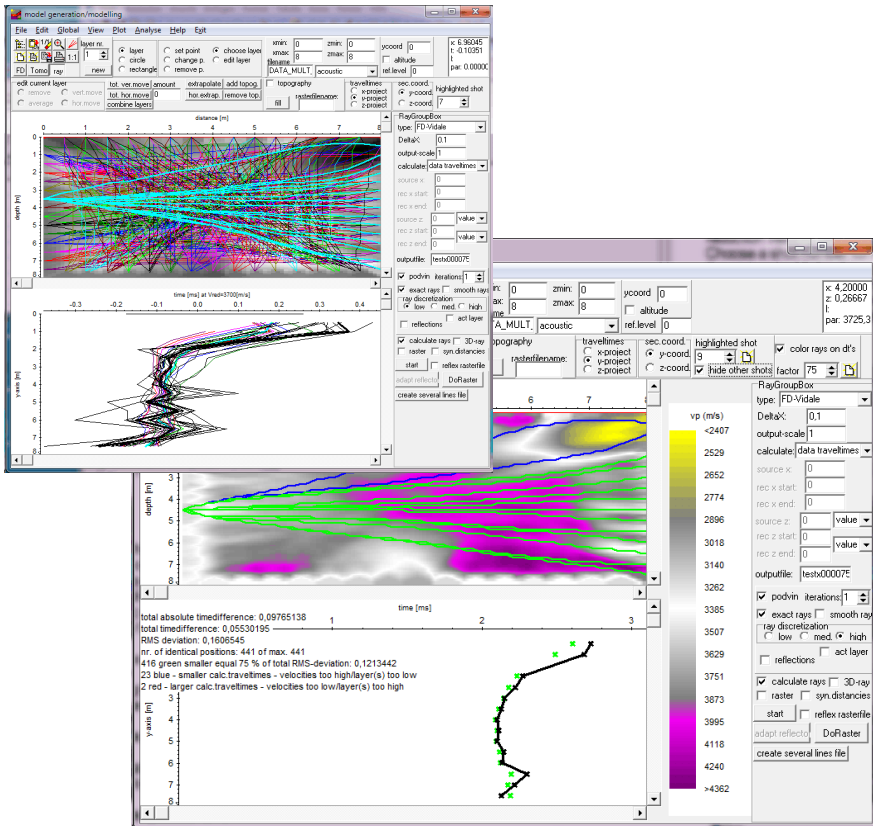


4. **Control** the tomographic result by a forward raytracing. For that purpose activate the option **ray**. The raytracing menu opens in addition. Load the traveltimes data using the option **File/load data traveltimes**. Then the screen is split vertically showing in the upper window the model together 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 or network..
- 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** - this must be done because otherwise the currently loaded model (normally the start model for the tomography) will be used for the raytracing
- 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. Within the **traveltimes** checkbox you must select over which axis the traveltimes data shall be plotted, e.g. for borehole-borehole data select y-project. You may use the reduction velocity plot option in order to stretch the time axis for a better visibility.

Choose a shot number for highlighting using the option **highlighted shot** (within the picture highlighted in cyan). You can interactively go through the highlighted shot numbers in order to analyse the differences of the individual shots. With the option hide other shots activated only the highlighted shot will be displayed.

You may also check for the mean traveltimes difference using the option **Analyse/calculate traveltimes differences using all coordinates** or **calculate traveltimes differences using actual projection**. The option **color ray** for dt's allows to colour the rays based on the actual traveltimes differences (green below rms deviation, blue for too small calculated traveltimes, red for too large calculated traveltimes).



0.10.4.3 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. 0.10.1) 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**

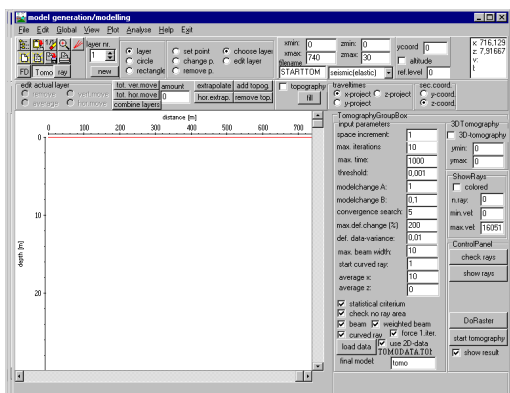
3. The **TomographyGroupBox** opens in addition (see figure on the right). Within this group box you have to enter the necessary tomography parameters.

- Load the data using the option **load data** (see also chap. 0.10.4.1). If the 3D-data format is used for the 2D-tomography 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. The ASCII-tomography data may be created within the modules

- 2D-dataanalysis - save picks using the formats ASCII-2D tomography or ASCII-3D tomography
- travelttime analysis - option export to ASCII
- modelling - export data traveltimes to ASCII

- 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.

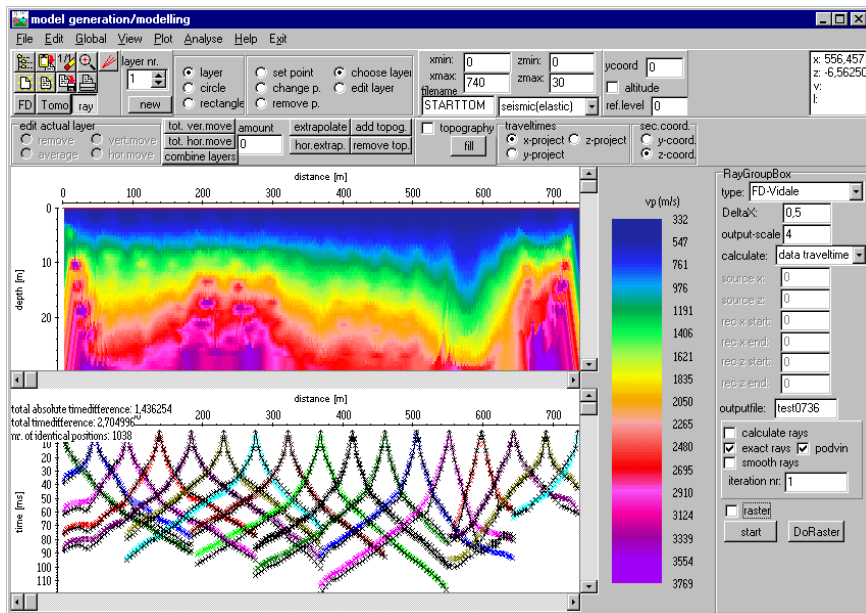
- The following options must be set for the refraction tomography:
 - activate the option **curved ray**.



- 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).
 - 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 traveltimes 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** - this must be done because otherwise the currently loaded model (normally the start model for the tomography) will be used for the raytracing
 - 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. Choose a shot number for highlighting using the option **highlighted shot**. You can interactively go through the highlighted shot numbers in order to analyse the

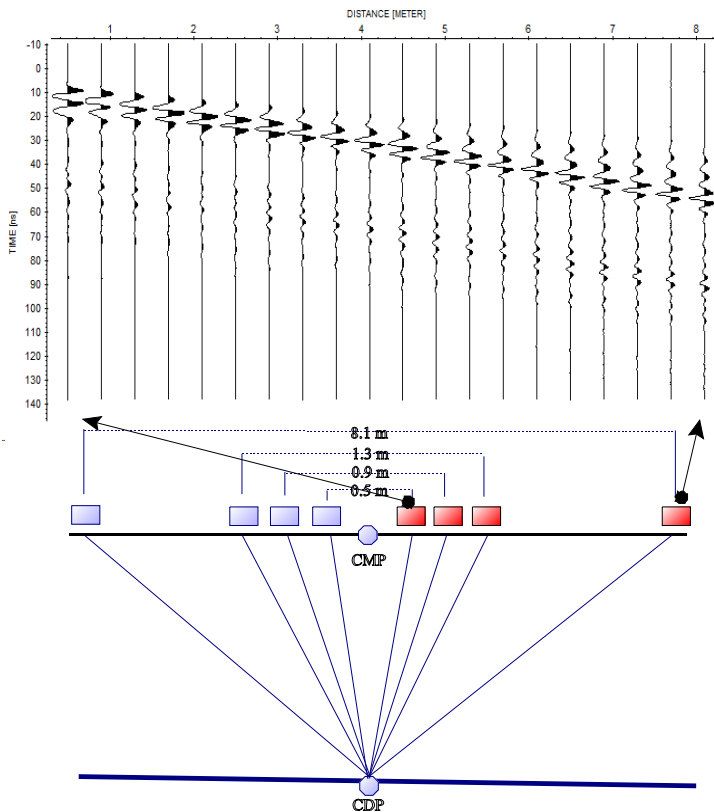
differences of the individual shots.

You may also check for the mean traveltimes using the option **Analyse/calculate traveltimes differences using actual coord.projection**



0.11 Use of CMP-analysis for a single or several independent CMP-files

In the following the use of the velocity adaptation for a single or several independent CMP-files is described. A single CMP data may be recorded along a zero-offset line at any position in order to get informations about the velocity distribution with depth. These informations may be used afterwards e.g. for performing a migration or a timedepth-conversion of the zero-offset line. Of course those measurements may also be performed as standalone measurements. The CMP measurement may be performed with moving source and receiver positions as shown below or by fixing the source and moving the receivers or using a receiver group. The import, processing and analysis are identical for all configurations.



0.11.1. Import the data data for single CMP

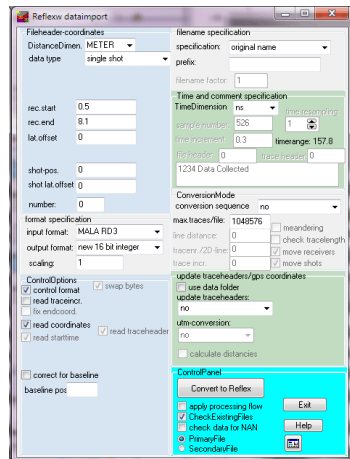
Enter the following parameters within the import menu:

- data type: single shot

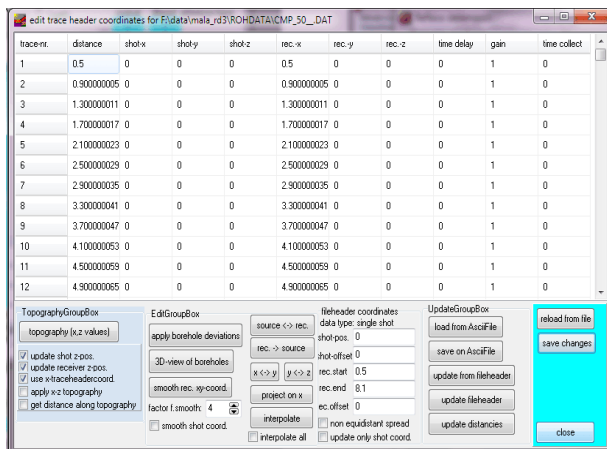
rec.start, rec.end and shot start must be entered as if the data had been acquired with a fixed shot position and a moving receiver. Therefore shot pos. corresponds to the CMP-position and rec.start "CMP-position plus the first shot-receiver separation" and rec.end "CMP position plus the last shot-receiver separation". Assuming CMP-position is given at 0 then the following values for the above example must be entered:

- rec.start: 0.5 m (first shot/receiver separation)
- rec.end: 8,1 m (last shot/receiver separation)

shot start: 0 m (position of the CMP)

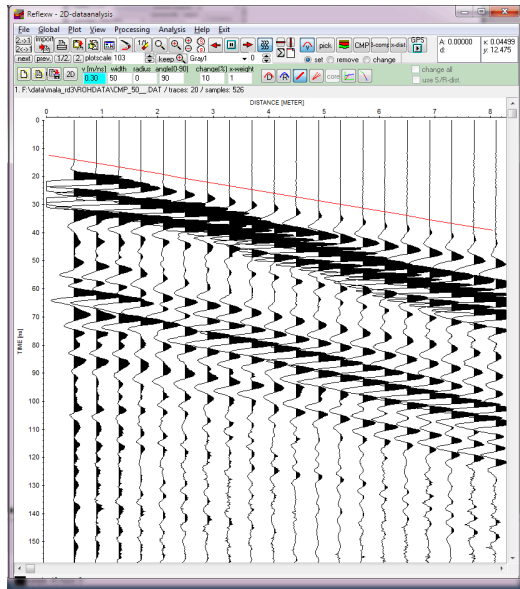


After having converted the data to Reflexw the edit traceheader menu opens. If the traceheader coordinates are not taken from the original data click on update from fileheader and save the changes. Close this window.

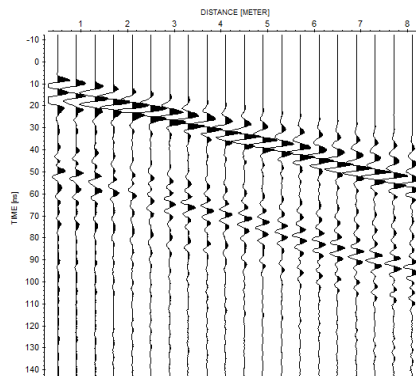


0.11.2. Processing single GPR CMP

Apart from other possible necessary processing steps the main problem with GPR CMP'S is given by the fact that the start time normally is not exactly given. Therefore the start time must be defined manually. If a direct air wave is present this direct air wave allows you to define the start time, this means the time the wave has been transmitted. Expand the visible distance range by the first shot/receiver separation using the plotoption manual scaling (in this case xmin has been set to 0, the other values had been kept) and enter the interactive velocity adaptation and choose the straight line adaptation. Choose a velocity of 0.3 m/ns and adapt the direct air. The crossing point at distance 0



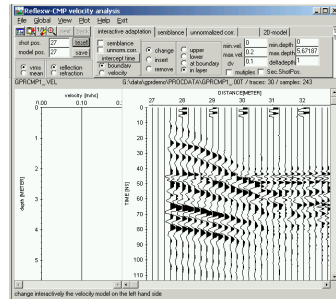
yields the time at which the wave has been transmitted - in this case 12 ns. This values now may be entered as a negative value within the fileheader. If no air wave is visible you may use the direct ground wave instead. Of course the velocity of the direct ground wave first



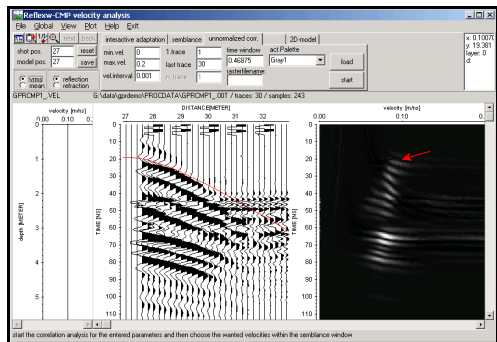
must be determined.

0.11.3. velocity analysis

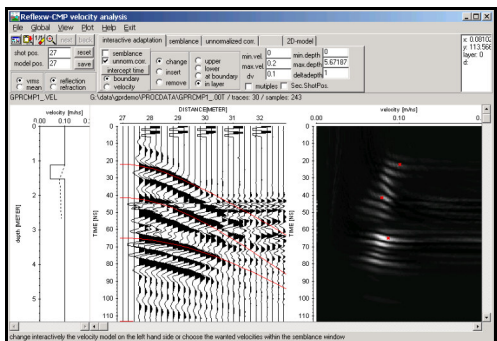
1. Enter the CMP-module and load the wanted cmp-file using the option **file/open**. The file is plotted together with a simple one layer startmodel on the left handside. By default the velocity for the first layer is set to 0.1 m/ns for GPR or 3000 m/s for seismics respectively. The default max. modeldepth is calculated from this velocity and the max. time of the loaded cmp-file.



2. activate the options **unnormalized cor.** or **semblance** and use for example the following input parameters for the min. and max. vel. and the vel.interval. Click on start and on the right hand side the correlation histogram is shown. Within this correlation histogram you may choose the best adapted velocities. Click on the chosen velocity within the correlation histogram and the current 1D-model will be updated. Choose the next best velocity for the next reflection and a second layer will be created and so on. The solid lines define the layer velocities, the dashed ones the mean (vrms) velocities. To be considered: the mean velocities correspond to those velocities which come from the interactive velocity adaptation within the 2D-dataanalysis. To be considered especially for GPR data: Only the first-breaks yield correct vrms and layer-velocities. If a later arrival (e.g. the second or third half-cycle) is used for the analysis too low vrms velocities result. Especially the semblance analysis may lead to these wrong velocities because the maximum semblance response normally relates to later arrivals



3. After having created all layers you still can change interactively (option **interactive adaptation** active) the layer boundaries and/or velocities by simply clicking on the boundary and drag it with pressed mouse key. The same is possible for the velocity within the layer (option velocity activated). A new layer boundary can be inserted (option insert activated) and an existing one can be removed (option remove activated). The current 1D-model can be stored using



any filename. The option model pos. defines the position of the 1D-model along the distance axis. This parameter is important for creating a subsequent 2D-model consisting of different 1D-models.

4. If several CMP's are present you may do the same procedure for each CMP. Then you may create a **2D-model** using the option 2D-model. Click on create and choose the wanted 1D-models. Enter the filename for the 2D-model. This 2D-model can be used for a subsequent 2D-migration or depth-conversion (option CMP-analysis within the 2D-velocity model groupbox). If only one CMP is present you still must create a 2D-model for the use within the migration or time-depth conversion. In this case you simply choose one 1D-model when creating the 2D-model. A laterally homogeneous 2D-model will be created.

0.12 Introduction to the use of 2D-velocity models for further processing

A 2D-velocity model will be used for different applications:

- migration
- time-depth conversion
- CMP-stacking
- NMO-correction

Depending on your data different methods for creating a 2D-velocity model are suitable.

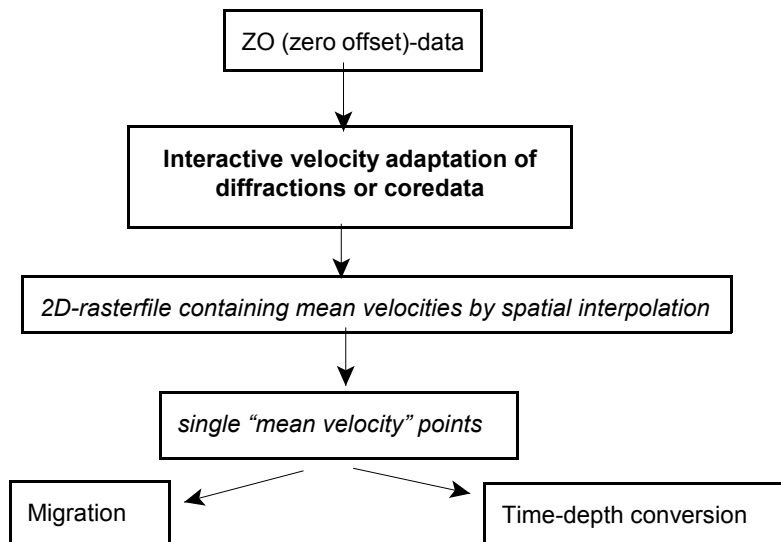
A layered model may be useful if distinct reflections are visible whereas a more smooth velocity model might be a good choice if only diffractions are present. It is possible to extract the velocity informations from the data to be processed (e.g. ZO data with diffractions for a subsequent migration or single shot data for a subsequent stacking) or from totally different data (e.g. refraction data).

Therefore different methods for creating a 2D-velocity field are implemented within Reflexw:

- interactive velocity analysis of zero offset data
- CMP velocity analysis
- layer picking within zero offset data
- interactive model generation based on seismic refraction data
- tomographic inversion

0.12.1. 2D-model from the interactive diffraction analysis

-> single mean velocity points from which a 2D-mean velocity model is created by spatial interpolation

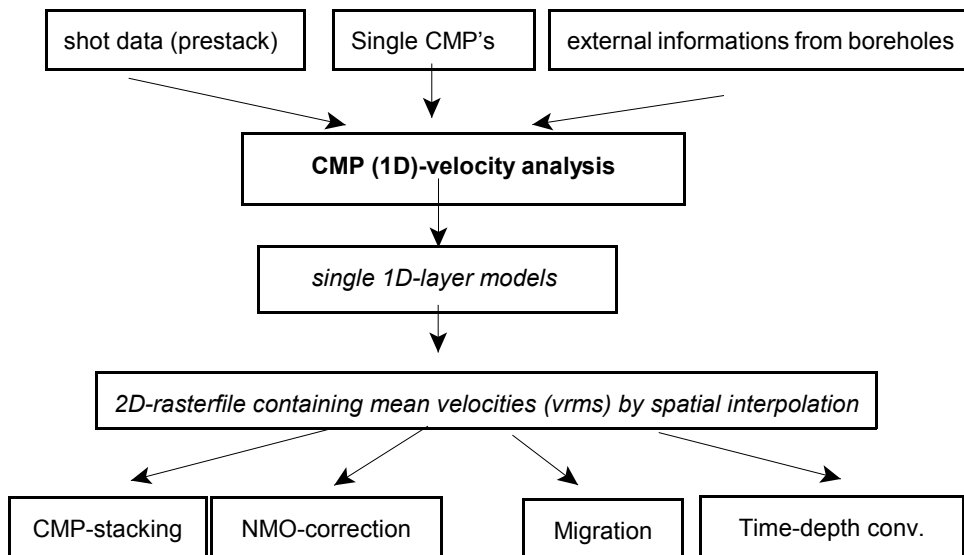


step by step guide:

- load the ZO-profile
- activate the option velocity adaptation
- choose and adapt the wanted diffractions or use the coredata adaptation
- save the adaptations on file
- click on 2D and enter a rasterfilename (optional)
- activate within the processing migration menu any of the 2D migrations or the timedepth conversion
- choose hyperb.adaptation (creates automatically a 2D-rasterfile) or rasterfile mean if you manually created a rasterfile using the option 2D
- perform the processing - the hyperb.adaptation file or the mean rasterfile will be queried.

0.12.2. 2D-model from CMP velocity analysis

-> single 1D-layer models from which a 2D layer velocity model is created by spatial interpolation



step by step guide for shot data (for a detailed description of the CMP-geometry and stacking see the seismic reflection guide):

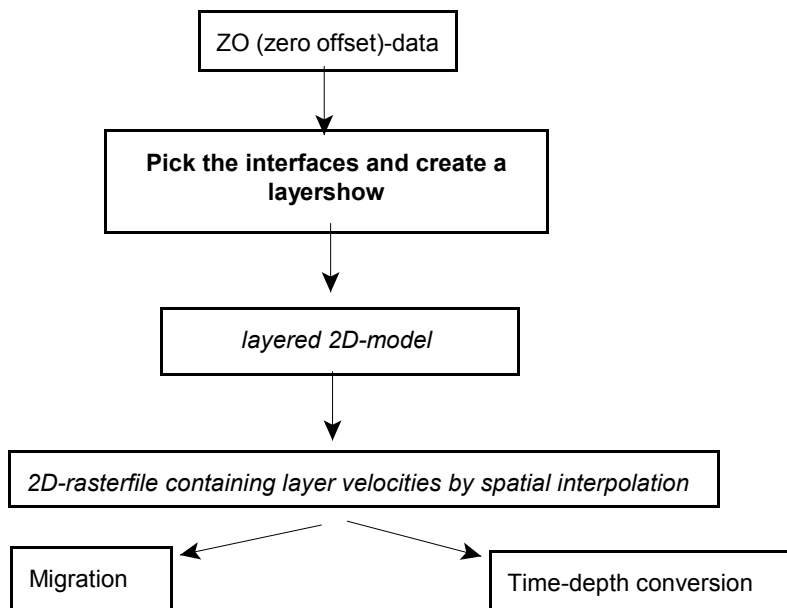
- load the single shot data (all shots must have been stored within one single file)
- activate the option CMP and choose the wanted ensembles for the velocity analysis
- enter the velocity analysis and create the 1D-models for the chosen ensembles
- enter the 2D-model panel and create the 2D-model (2dm-file). If a rasterfilename has been entered this Reflexw rasterfile can be used later for stacking
- close the velocity analysis menu and activate load 2D-model. Activate for filetype 2D-models if you want to load the created 2D-model (spatial interpolation will be done automatically). The filetype Reflexw rasterfile mean allows you to load the already rastered mean velocityfile when creating the 2D-model.
- perform the stacking or the nmo correction

step by step guide for ZO-data and additional single CMP-data

- enter the CMP(1D)-velocity analysis menu load the single CMP data
- create the 1D-model for the choosen CMP and enter the shot and model position
- load the next CMP and create the next 1D-model together with the correct shot and model position
- enter the 2D-model panel and create the 2D-model (2dm-file). If a rasterfilename has been entered this Reflexw rasterfile can be used later for migration or timedepth conversion
- close the velocity analysis menu and enter the 2D-analysis module
- activate within the processing migration menu any of the 2D migrations or the timedepth conversion
- choose CMP-analysis (creates automatically a 2D-rasterfile) or rasterfile mean for the already rastered mean velocityfile when creating the 2D-model.
- perform the processing - the hyperb.adaptation file or the mean rasterfile will be queried.

0.12.3. 2D-model from the layershow (picked interfaces)

-> Reflexw layer rasterfile

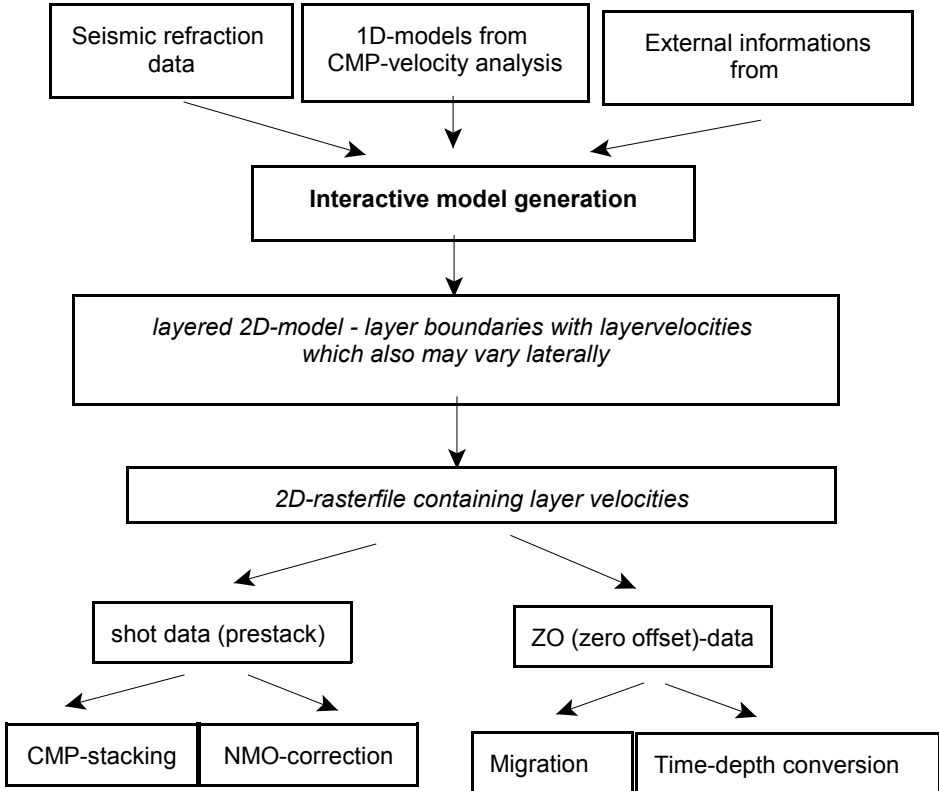


step by step guide:

- load the ZO data within the 2D-dataanalysis
- activate pick and pick the interfaces and store each interface on a separate file with the velocity above the interface (layernumber must increase with depth)
- pick and save a last horizontal layer at the max. traveltime or a little bit less by setting one pick at the profilestart and another pick at the profileend and use the option interpolate (the interface will be continuous). Use for velocity the velocity of the lowermost layer
- enter the layershow and create the layershow using the pickfiles (option layer pick vel. within the velocity choice panel must be chosen).
- click on export and activate the option generate Reflexw rasterfile, click on start
- close the layershow panel and activate within the processing migration menu any of the 2D migrations or the timedepth conversion
- choose rasterfile layer for the already rastered layer velocityfile
- perform the processing - the layer rasterfile will be queried. menu

0.12.4. 2D-model from the modelling menu, e.g. from refraction data

-> Reflexw layer rasterfile which will be used for a different dataset like a zero offset file for subsequent processing



step by step guide:

- enter the model generation menu
- create manually a 2D-velocity depth model or generate the model from seismic refraction data or from single 1D-models
- enter a rasterfilename and activate the option fill
- close the model generation menu and load the different dataset (either ZO-data or single shot data)

single shot data:

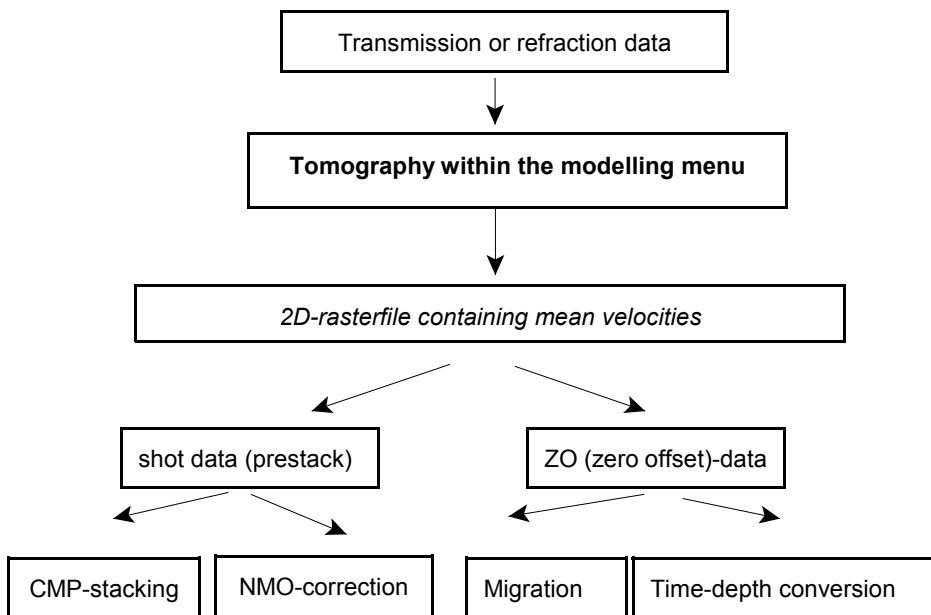
- load the single shot data and activate the option CMP
- choose the wanted ensembles for the stacking
- activate load 2D-model - choose for filetype Reflexw rasterfile layer for the already rastered layer velocityfile
- perform the stacking or the nmo correction

zero offset data:

- load zero offset data and activate within the processing migration menu any of the 2D migrations or the timedepth conversion
- choose rasterfile layer for the already rastered layer velocityfile
- perform the processing - the layer rasterfile will be queried

0.12.5. 2D-model from tomography

-> Reflexw mean rasterfile which will be used for a different dataset like a zero offset file for subsequent processing



step by step guide:

- enter the model generation menu
- perform a transmission or refraction tomography
- close the model generation menu and load the different dataset (either ZO-data or single shot data)

single shot data:

- load the single shot data and activate the option CMP
- choose the wanted ensembles for the stacking
- activate load 2D-model - choose for filetype Reflexw rasterfile mean for the tomographic model
- perform the stacking or the nmo correction

zero offset data:

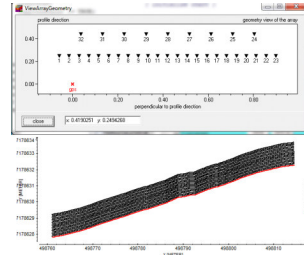
- load zero offset data and activate within the processing migration menu any of the 2D migrations or the timedepth conversion
- choose rasterfile mean for the tomographic model
- perform the processing - the mean rasterfile will be queried

0.13 GPS based Radar array data

REFLEXW allows to handle GPR array data from different manufacturers as:

- MIRA from Mala
- Stream from IDS
- Utsi
- Raptor from Impulse Radar
- 3D Radar

The GPR device normally consists of different transmitter and receiver antenna which may be combined in different ways. In most cases the geometry of the array is stored within an ASCII file. The data acquisition is normally done based on a single GPS mounted on the GPR device. Based on these gps coordinates and the relative array coordinates the position of each channel may be calculated. This is done automatically during the import of the data within Reflexw.



The following files must be present for the different devices:

MIRA from Mala:

the data are either stored on single 2D-lines or multiplexed within one single file.

rad and rd3- datafile of each channel or one single file for all channels

CHANNELS:24

CHANNEL CONFIGURATION:T1 - R1

cfb - geometry file of the array for the whole array (filename as datafilename (multiplexed data) or without 5 last characters (single 2D-lines))

1 T1R1 1.270000 -0.454000 1

2 T2R1 1.190000 -0.454000 1

rad - contains the geometry for the whole array or if all channels are stored within one single rd3-file

NUMBER_OF_CH:16

CH_X_OFFSETS:0.078 0.158 0.238 0.318 0.398 0.478 0.558 0.638 0.718 0.798 0.878 0.958 1.038 1.118 1.198 1.278

CH_Y_OFFSETS:3.013 3.013 3.013 3.013 3.013 3.013 3.013 3.013 3.013 3.013 3.013 3.013 3.013 3.013 3.013 3.013

utm - utm file for the whole array (filename as datafilename (multiplexed data) or without 5 last characters (single 2D-lines))

tracnumber	converted latitude	converted longitude	elevation	utm-zone
------------	--------------------	---------------------	-----------	----------

2	7607031.67	561559.74		8.33
---	------------	-----------	--	------

14	7607031.67	561559.75		8.33
----	------------	-----------	--	------

pos - gps or local coordinate file for the whole array (filename as datafilename (multiplexed data) or without 5 last characters (single 2D-lines))

tracnumber (converted)	latitude	(converted) longitude	elevation
1	6611453.473800	650087.201400	5.755800
9	6611453.554100	650086.624000	5.750200

IDS Stream C:

dt - datafile of each channel containing the relative geometry within the array

GPS_config.txt - file containing the position of the GPS-device relative to the antenna centre

```
<OFFSET_GPS_ARRAY_XYZ_METERS>
0.42,0.13,0
```

gec - xyz local or utm coordinates for the whole swath

```
<VERSION>
07.00.000
<OFFSET_GPS_ARRAY_XYZ>
0.13,0.42,0
<UTM_ZONE>
32,1
<REFERENCE_XYZ> converted longitude, converted latitude
512941.685563995,5290630.10008273,433.657
<REFERENCE_LLA>
0,0,0
<XCELL>
0.0416389
<VALID_MARKERS_SWEEP>
69
1,0,0,512941.685563995,5290630.10008273,433.657,32,1
8,0,1.599999999999854,512941.370795237,5290630.019358,433.667,32,1
```

geox - gps file for each channel

```
<01.01.000>
<XYZLatLonAltTL>
764
1,-.331284451472892,.39801794668614,1,0,0,.517849083876708
2,-.372188774882243,.396415705932969,1,0,0,4.09356916273305E-02
3,-.415734322142434,.395758968497921,1,0,0,4.35504993111233E-02
4,-.46195840176718,.395337384737874,1,0,0,4.62260020986197E-02
5,-.510303558617004,.394877267426637,1,0,0,4.83473463467663E-02
6,-.558162396879843,.393745335548008,1,0,0,4.78722223178161E-02
7,-.604604194717226,.391815503848043,1,0,0,.046481876433278
```

gps - gps coordinates for the whole swath, the synchronization with the radar data works using markers within the dt-file. It is not recommended to use this file if the warning message appears that the number of markers and the number of gps coordinates do not coincide.

Raptor from Impulse Radar:

iprb and iprh - datafile of each channel, iprh also contains channel geometry

CHANNEL CONFIGURATION: T1 - R1

CH_X_OFFSET: -0.710000

CH_Y_OFFSET: 0.000000

ord - geometry file of the array (filename without 4 last characters)

0 -0.710000 0.000000 1

1 -0.630000 0.000000 1

cor - gps file for the whole array (filename without 4 last characters)

tracer	date	time	latitude	longitude	elevation				
3	2018-05-11	09:38:40:796	64.73240969383	20.97510585233	E	11.665	M	4	
6	2018-05-11	09:38:41:203	64.73240943933	20.97510009633	E	11.646	M	4	

Utsi:

gps - contains the gps coordinates for each file - identical for each channel

dat - datafile

gpt - gps synchronization file

3D-Radar:

nav - contains the navigation data as well as the channel geometry

```
<AntennaElementInfos>
  <AntennaElementInfo TransverseOffset="-0.75" RelativeScanTime="0" TxRxDistanceX="-0.44"
TxRxDistanceY="0.074999999999999956" TxRxDistanceZ="0" TxElementNumber="0" RxElementNumber="0" />
  <AntennaElementInfo TransverseOffset="-0.675" RelativeScanTime="0" TxRxDistanceX="-0.44"
TxRxDistanceY="-0.0750000000000000067" TxRxDistanceZ="0" TxElementNumber="1" RxElementNumber="0" />
  <AntennaElementInfo TransverseOffset="-0.6" RelativeScanTime="0" TxRxDistanceX="-0.44"
TxRxDistanceY="0.074999999999999956" TxRxDistanceZ="0" TxElementNumber="1" RxElementNumber="1" />
```

```
<TriggerPosition Lat="51.357426374537489" Lon="-1.0838945838397325"
Velocity="0.01567959226667881" Heading="98.008865356445313" DHeading="-
0.0031359302811324596" SourceType="Normal" />
```


0.13.1. Import the data

The import from the different array data formats is performed in a similar way. In each case the resulting Reflexw files are 2D-files each containing the corrected (based on the antenna geometry) gps, utm or local coordinates stored within the traceheaders.

The standard automatic import procedure is the following:

1. Convert the original data to 2D-lines
2. Load the gps-data from an external file (one file for the whole swath)
3. Correct the gps-data depending on the antenna position relative to the GPS device

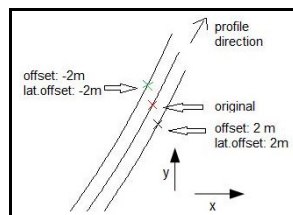
This procedure and the necessary settings are described below in detail.

The **conversion sequence** must be set to **multichannel** (any other conversion sequence like several lines is also possible but only in this case the offsets of the antenna are read from the original data). This allows to import a set of 2D-lines or swaths within one step. It is even possible to import files of different antenna setups if the antenna array geometry is read from the original data (option use correctoffset.txt deactivated). In addition it is possible to set an overall swath offset and antenna offset within the swath. These values are used for an update of the fileheader coordinates which also may serve as the coordinate base if no gps coordinates are present.

For the import of the gps coordinates you must choose the corresponding **update**

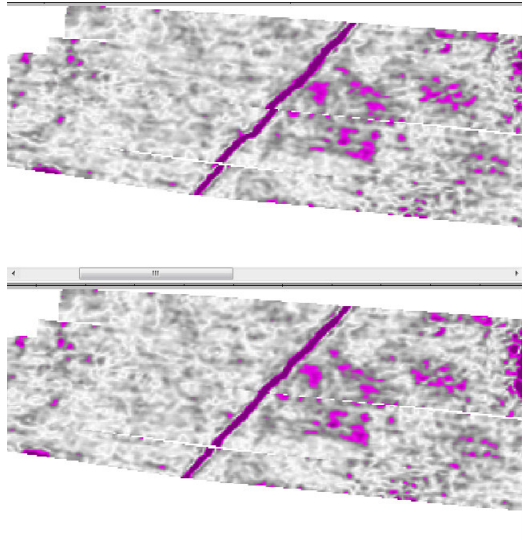
traceheader format. This depends on your acquisition system (see below). Two parameters must be entered correctly. The option **1. original coord.** allows you to specify where the first original coordinates shall be stored - either on the x- or y-traceheader coordinates. The option **latitude** allows to tell where the latitude has been stored within the traceheader depending on the setting of the 1. original coord. (latitude in degrees either in x- (shot x, rec x and CMP x-pos) or y-coord). The longitude is automatically saved vice versa in y- or x-coord. It is recommended to always save the latitude or converted latitude values on the y-coordinate. In most cases both the original utm and the gps coordinates have a format where the latitude coordinates are saved under the 1. original position. In this case set both 1. original coord. and latitude to y-coord.

The correction of the antenna array geometry in relation to the gps-device is done by activating the option **correct for offset**. Precondition is that the coordinates are in meter. Therefore a utm-conversion must be done if the gps-coordinates are given in degrees (set utm-conversion to UTM-conversion).



With activated option use correctoffset.txt or if the conversion sequence has not been set to multichannel the existing correctoffset.txt which must have been defined before manually is used for the correction of the antenna array instead of the original data. If deactivated the original geometry is read in and the correctoffset.txt is always automatically updated based on the original array coordinates.

A good **check** for the effectiveness of the antenna geometry correction may be done by analysing the resulting timeslices and looking for linear features which are not orientated in profile direction. If these features show distinct steps the correction failed. The right picture shows such a wrong correction because the position of the gps-antenna in profile direction was wrongly entered. The lower picture shows the correct result with a corrected position of the gps device in profile direction.



Following the parameter settings for the the different data acquisition systems are listed.

0.13.2. Mira from Mala

The original Mira data are either saved as single 2D-lines or multiplexed within one single file containing all data of the antenna array. Multiplexed means that the sorting of the data is based on the channels (1. Trace of all channels, then 2. trace of all channels and so on). In this case Reflexw automatically performs a demultiplexing during the import and stores the files as individual 2D-lines with a filename extension indicating the channel number (number defines the naming value of the first channel). If single 2D-lines are present the filename contains both the channel (last 2 characters) and the swath number (last characters 6-8).

The Mira system generates either utm or local coordinates or degree data. For Utm (file extension utm) or local coordinates or degree data given in decimal(file extension pos) use as **update traceheaders WSKTRANS-GPS** which format fits in most cases. If a cor-file is present use **RAMAC-GPS**. Here the gps-data are given in a special own format.

The option **correct for offset** allows to correct the offsets of the antenna in relation to the gps-device. With activated option **“use correctoffset.txt”** the values within the existing correctoffset.txt file are used for the correction of the antenna array instead of the original data. If deactivated the original geometry is read in: if the original data are multiplexed 3D-data - this means all data of the antenna array are saved within one single file - the antenna geometry is read from the rad-file. Otherwise the offsets (perpendicular to and in profile direction) are read from a file with the extension cfb. The original offsets are defined as: offset in profile direction will be added to the gps coordinate, lateral offset perpendicular to the profile direction is assumed to be positive right to the profile direction and vice versa. The file correctoffset.txt will be updated based on these geometries. To be considered: the columns within the resulting correctoffset.txt file are defined different from the columns within the cfb-file. Here the first column defines the offset in profile direction and the second column the offset perpendicular to it - see example below. In addition the profile direction offset has been defined in opposite direction to that within the cfb-file.

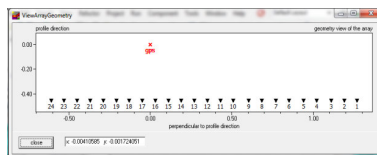
Example of four channels from 24 channels in total:

original cfb-file

```
1 T1R1 1.270000 -0.454000 1
2 T2R1 1.190000 -0.454000 1
1 T2R2 1.110000 -0.454000 1
2 T3R2 1.020000 -0.454000 1
```

resulting correctoffset.txt file

```
1 0.4540000000000000 1.2700000000000000
2 0.4540000000000000 1.1900000000000000
3 0.4540000000000000 1.1100000000000000
4 0.4540000000000000 1.0200000000000000
```



A very simple **processing flow** for the whole dataset is shown on the right side.

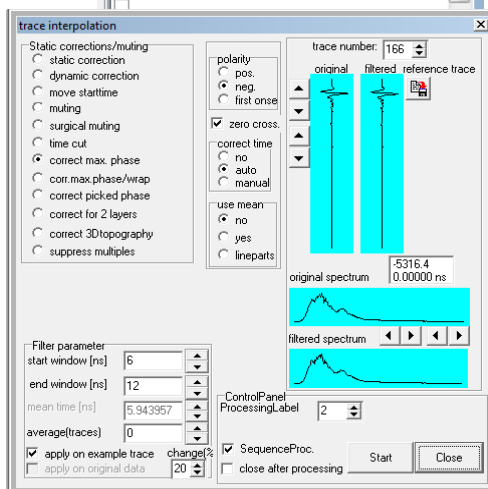
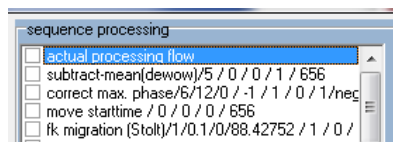
The single channels normally exhibit different time zero values.

The time zero correction may be done using the correct max. phase option after application of the subtract

mean(dewow) filter. The parameters shown on the right side often satisfy the needs for a stable correction. Nevertheless this should be controlled.

Besides these processing steps standard processing steps like gain, bandpassfiltering, migration, background removal, may be applied.

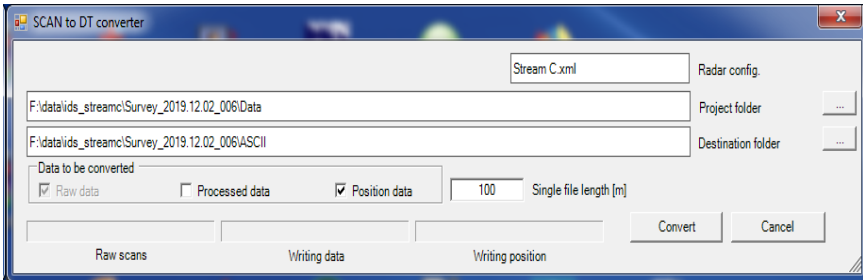
The option normalize profiles is sometimes necessary because of possible different amplitude distribution of the individual antenna channels.



0.13.3. IDS Stream C

Reflexw only allows to import dt-files. These files must be generated using the IDS HD-data converter.

Within this data converter the option Position data must be activated. The resulting dt-files will be stored within the destination folder. It is recommended to use the import folder of Reflexw as destination folder.



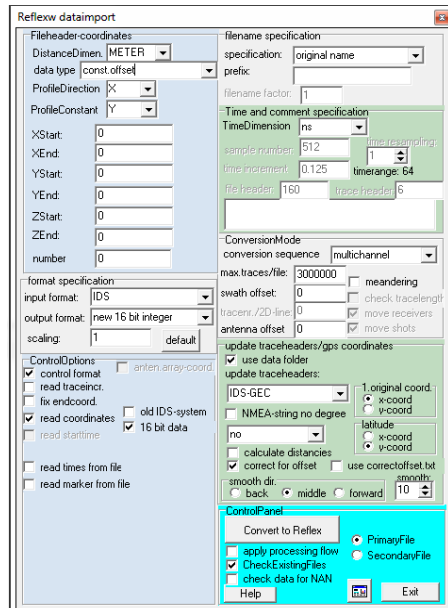
The original IDS dt data are saved as single 2D-lines. The filename contains both the channel ((characters 3 and 4) and the swath number (characters 5-8).

The HD-data converter program generates **dt-files** containing the radar data and the relative coordinates of the antenna and **gps** files containing the gps and **gec** files containing the utm- or local coordinates. It is recommended to use the gec-files because the gps-data are often much more irregular and often the number of markers does not coincide with the number of gps lines within the gps-file.

The option **read coordinates** must be activated (the option old IDS-system should be deactivated for almost all systems).

For **gec** files use the **IDS-GEC** as **update traceheaders**. Both the gps-file and the gec-file contain the latitude and the converted latitude values within the 2. place in contrast to most of the other systems. Therefore it is recommended to set 1. original const. To x-coord. and latitude to y-coord.

The option **correct for offset** allows to correct the offsets of the antenna in relation to the gps-device. With activated option "**use correctoffset.txt**" the values within the existing correctoffset.txt file are used for the correction of the antenna positions instead of the original data. If deactivated the original geometry is read in from the dt-files (relative antenna positions) and from the the



GPS_config.txt file which contains the relative coordinates of the gps-device. The original offsets are defined as: offset in profile direction will be subtracted from the gps coordinate in profile direction, lateral offset perpendicular to the profile direction is assumed to be positive left to the profile direction and vice versa and will be subtracted from the gps coordinate in profile constant direction. The file correctoffset.txt will be updated based on these geometries. To be considered: the columns within the resulting correctoffset.txt file are defined different from the columns within the gps-config.txt file. Here the first column defines the offset in profile direction and the second column the offset perpendicular to it - see example below.

Example of four channels from 32 channels in total:

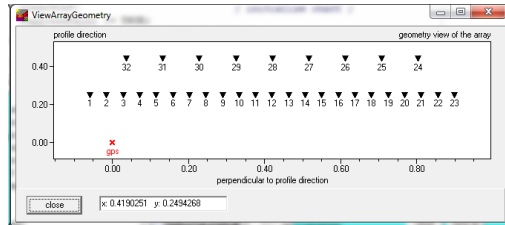
original coordinates within dt-file

```
1 0.4785 0.38
2 0.4350 0.38
3 0.3915 0.38
4 0.3480 0.38
```

.....

Gps-file

```
0.42 0.13
```



Offset in profile direction for channel 1: $0.13 - 0.38 = -0.25$

Lateral offset for channel 1: $0.42 - 0.4785 = -0.0585$

resulting correctoffset.txt file

```
1 -0.250000 -0.058500
2 -0.250000 -0.015000
3 -0.250000 0.028500
4 -0.250000 0.072000
```

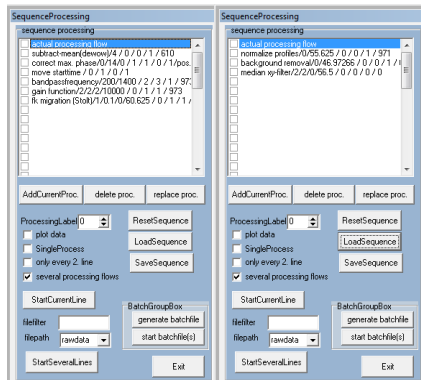
.....

The IDS Stream C consists of 23 VV dipoles and 9 HH dipoles. You may create a 3D-file only from the VV dipoles or HH dipoles or from the total dataset.

Depending on this choice you may adapt your **processing flow**.

A useful processing flow for the whole dataset including the VV and HH dipoles is shown on the right side.

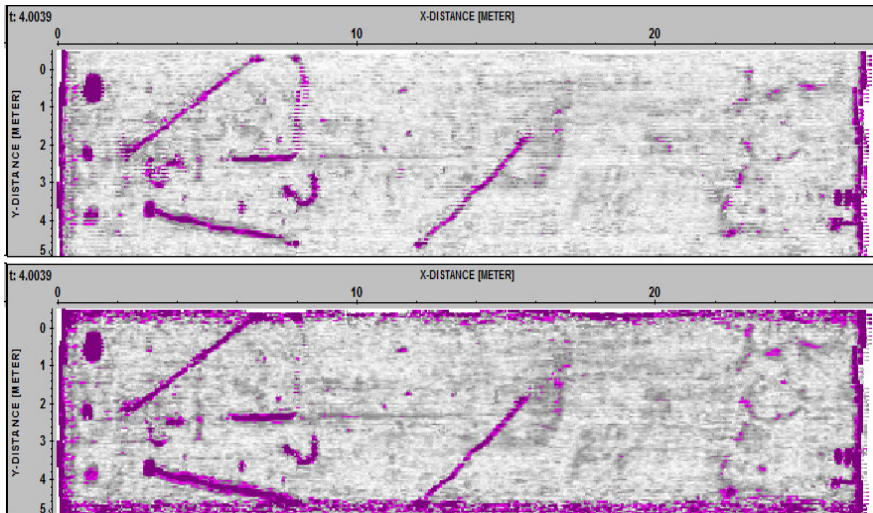
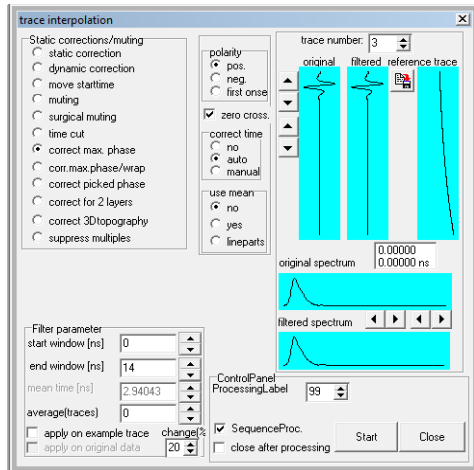
It has been subdivided into two individual processing flows which may be performed within one step by activating the option several processing flows. Such a subdivision is necessary because the option normalize profiles within the second processing flow must stand at the first position of the processing flow and it makes no sense to use this option for the very first step.



The single channels normally exhibit different time zero values and sometimes different overall amplitude distributions.

The time zero correction may be done using the correct max. phase option after application of the subtract mean(dewow) filter. The parameters shown on the right side often satisfy the needs for a stable correction. Nevertheless this should be controlled.

The option normalize profiles is sometimes necessary because of the different amplitude distribution of the individual antenna channels. If this option is not used the resulting timeslices may show significant stripes especially when using both the VV and HH dipole data (see picture below - upper panel). Depending on the data the option normalize profiles may be performed before the optional application of the background removal or afterwards.



3D-data generation:

The 3D-data generation is very simple. Use as type of interpolation use interpolation scheme for freely distributed 2D-lines and for example as X- and Y-rasterincrement 0.05 m and and the double values for the interpolation. These are useful values because of the offsets between the dipoles of 8 cm within the Stream C array and the small interpolation allows a full coverage if the data are not acquired strictly along parallel lines. As sorting receiver coordinates must be entered.

Generate 3D-file from 2D-lines

3D-filename: file_3da

type of interpolation: use interpolation scheme for freely distributed 2D-lines

☐ add x-y-scans by choosing the max. abs. value

☒ apply mean filter in time direction 1

3D coordinates

XStart: 0

XEnd: 27.5

YStart: -0.5

YEnd: 5

XRasterincrement: 0.05

YRasterincrement: 0.05

XInterpolation: 0.1

YInterpolation: 0.1

interpol.weight: linear weight

timerange/sorting

time end: 30

timeincrement scale: 1

sorting: receiver coordinates

2D-files choice

filepath: procddata

filefilter: *.12t

load 2D-files: LA010001.12T
LA010002.12T
LA010003.12T

min/max.xy coord. LA010003.12T

☒ update traceheader coordinates

data scaling: 1

Start Cancel

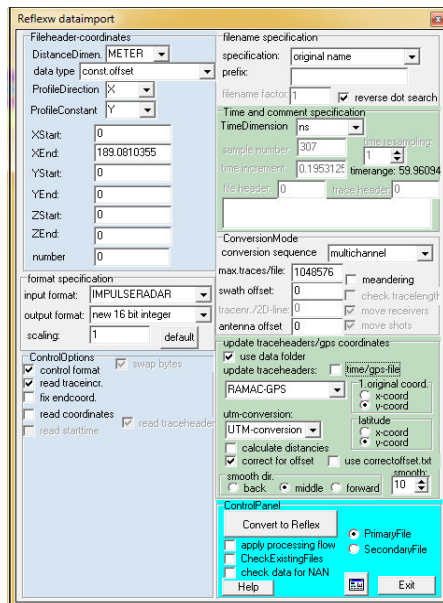
0.13.4. ImpulseRadar (Raptor)

The original Raptor data are either saved as single 2D-lines or multiplexed within one single file containing all data of the antenna array. Multiplexed means that the sorting of the data is based on the channels (1. Trace of all channels, then 2. trace of all channels and so on). In this case Reflexw automatically performs a demultiplexing during the import and stores the files as individual 2D-lines with a filename extension indicating the channel number (number defines the naming value of the first channel). If single 2D-lines are present the filename contains both the channel (last 2 characters) and the swath number (last characters 5-7). The 2D-files have the extension iprb and iprh, the multiplexed file has the extension ipr and iph.

There exist different types of gps-data:

- cor-file:

If the raptor system has generated a cor-file which contains degree data use **RAMAC-GPS** as update traceheaders type and in most cases 1. original coord. set to y-coord and latitude set to y-coord..The cor-filename is the same as for the ipr file and as for iprb files except the last 4 characters indicating the channel. For the iprb file only one cor-file is given for all channels. A possible latency must be included afterwards by using the option correct for offset within the edit traceheader menu or within the edit several fileheaders menu. The gps-data are given as in a special own format with the first column indicating the tracenummer following with the date and time and the gps coordinates.



Filename examples:

5_9_2023 104922 AM_004_A01.iprb
5_9_2023 104922 AM_004_A01.iprh
5_9_2023 104922 AM_004.cor

cor-file

1	2023-06-07	08:54:15:600	49.74780606667	N	10.20359405750	E	203.836	M	4
3	2023-06-07	08:55:22:400	49.74780558033	N	10.20359243800	E	203.835	M	4

- time and gps file:

The newer Raptor acquisition software does not generate a cor-file anymore but only a time and a gps-file. The time/gps filenames are the same as for the ipr file and as for iprb files except the last 4 characters indicating the channel. If these files are present use again **RAMAC-GPS** as update traceheaders type together with activated option time/gps-file and in most cases set 1. original coord. to x-coord and latitude set to y-coord. This is different to the cor file because normally the columns containing the longitude and latitude within the gps file are exchanged in comparison to the cor-file.

Based on the time and the gps file a gpss-file is generated containing the tracenummer within the first column in addition. A possible latency must be included afterwards by using the option correct for offset within the edit traceheader menu or within the edit several fileheaders menu.

Filename examples:

5_9_2023 104922 AM_004_A01.iprb

5_9_2023 104922 AM_004_A01.iprh

5_9_2023 104922 AM_004.time

5_9_2023 104922 AM_004.gps

gps-file

2023-06-07	08:54:15:600	10.2035940575	49.74780606666666	203.8364	0.6	17
------------	--------------	---------------	-------------------	----------	-----	----

2023-06-07	08:55:22:200	10.2035933145	49.74780581516667	203.8344	0.6	17
------------	--------------	---------------	-------------------	----------	-----	----

time-file

1	2023-06-07	08:55:22:081
---	------------	--------------

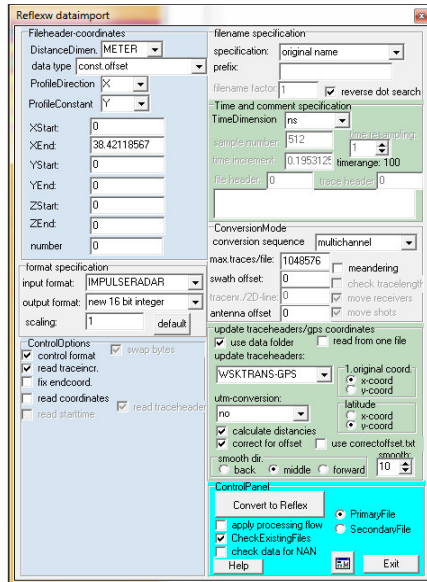
2	2023-06-07	08:55:22:237
---	------------	--------------

resulting gpss-file

1	2023-06-07	08:54:15:600	10.2035940575	49.74780606666666	203.836	4	0.6	17
2	2023-06-07	08:55:22:200	10.2035933145	49.74780581516667	203.834	4	0.6	17

- pos file:

When using the ViewR program from ImpulseRadar in order to export the data the GPR data are multiplexed data containing all channel data within one file. The filename extensions are ipr and iph. In this case a pos file is generated which normally contains utm coordinates. The filename is the same as for the gpr data including _G01 and the extension pos. Use as update traceheader format WSKTRANS-GPS. and in most cases set 1. original coord. to x-coord and latitude set to y-coord. If the coordinates are given in meter a utm conversion is not necessary. The pos file includes the x-, y-, z-coordinates which are all saved within the Reflexw traceheaders. As the time is not given a change based on a possible latency cannot be done afterwards. Such a correction might have already been done within the ViewR program.



Filename examples:

5_9_2023 104922 AM_004.iph
 5_9_2023 104922 AM_004.ipr
 5_9_2023 104922 AM_004_G01.pos

pos-file

2	352268.523213	5745966.681008	44.350
3	352268.452970	5745966.683447	44.335

- tsp file, Raptor 45 with total station GPS

The Raptor 45 with total station GPS generates a different GPS ASCII-file with the extension tsp. gps-file. The tsp filenames is the same as for the ipr file and as for iprb files except the last 4 characters indicating the channel. Use as update traceheader format WSKTRANS-GPS. Normally the GPS-data are given in METER. Therefore no utm-conversion is necessary. The 1. Original coord. and the latitude must be set according to the original settings.

The tsp file includes the x-, y-, z-coordinates as well as the time which are all saved within the Reflexw traceheaders. A possible latency must be included afterwards by using the option correct for offset within the edit traceheader menu or within the edit several fileheaders menu.

Filename examples:

Braunschweig_001_A01.iprb

Braunschweig_001_A01.iprh

Braunschweig_001.tsp

tsp-file

3	5792124.700000	32603203.340000	75.270000	10:13:18:100	0
6	5792124.700000	32603203.330000	75.270000	10:13:18:300	0

To be considered for all gps formats:

The columns within the different gps files containing the longitued lateitude values or the converted utm x-, y-coordinates have different sortings (exchanged columns) within the different files. Therefore it is recommended to always control the correct setting of the option 1. original.coord.

The option **correct for offset** allows to correct the offsets of the antenna in relation to the gps-device. Precondition is that the coordinates are in meter. Therefore a **utm-conversion** must be done if the gps-coordinates are given in degrees (set utm-conversion to UTM-conversion).

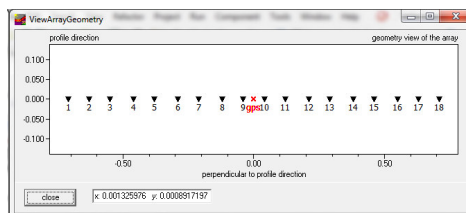
With activated option "**use correctoffset.txt**" the values within the existing correctoffset.txt file are used for the correction of the antenna array instead of the original data. If deactivated the offsets (perpendicular to and in profile direction) are read from a file with the extension ord if the original database are 2D-lines or from the idh file in case of a multiplexed 3D-file. The original offsets are defined as: offset in profile direction will be added to the gps coordinate, lateral offset perpendicular to the profile direction is assumed to be positive right to the profile direction and vice versa. The file correctoffset.txt will be updated based on these geometries. To be considered: the columns within the resulting correctoffset.txt file are defined different from the columns within the ord-file. Here the first column defines the offset in profile direction and the second column the offset perpendicular to it - see example below. In addition the profile direction offset has been defined in opposite direction to that within the ord-file. Example of four channels from 18 channels in total:

original ord file

```
0 -0.710000 0.000000 1
1 -0.630000 0.000000 1
2 -0.550000 0.000000 1
3 -0.460000 0.000000 1
```

resulting correctoffset.txt file

```
1 -0.000000 -0.710000
2 -0.000000 -0.630000
3 -0.000000 -0.550000
4 -0.000000 -0.460000
```



In case of ipr/iph data the iph file contains the following informations:

CH_X_OFFSETS:-0.294 -0.210 -0.126 -0.042 0.042 0.126 0.210 0.294

CH_Y_OFFSETS:0.100 0.100 0.100 0.100 0.100 0.100 0.100 0.100

-> resulting correctoffset.txt file:

```
1 0.100 -0.294
2 0.100 -0.210
3 0.100 -0.126
4 0.100 -0.042
5 0.100 0.042
6 0.100 0.126
7 0.100 0.210
8 0.100 0.294
```

A possible processing flow for the whole dataset is shown on the right side.

The single channels normally exhibit different time zero values.

The time zero correction may be done using the correct max. phase option after application of the subtract mean(dewow) filter. The parameters shown on the right side often satisfy the needs for a stable correction. Nevertheless this should be controlled.

Besides these processing steps standard processing steps like gain, bandpassfiltering, migration, background removal, may be applied.

The option normalize profiles is sometimes necessary because of possible different amplitude distribution of the individual antenna channels.

The image shows two overlapping software windows. The top window, titled 'SequenceProcessing', contains a list of processing steps with checkboxes: 'actual processing flow', 'subtract-mean(dewow)/5 / 0 / 0 / 1 / 745', 'correct max. phase/5000/450 / 0 / 1 / 1 / 1/a', 'move starttime / 0 / 1 / 1 / 1', 'gain function/2.654208/1/4.226892/10000 / 0', 'bandpassfrequency/120.4706/888.4706 / 2 /', 'background removal/0/71.67969 / 0 / 0 / 1 / 1', and 'lk migration (Stoll)/1/0.175/0/71.67969 / 0 / 0'. The bottom window, titled 'trace interpolation', has several sections. On the left, 'Static corrections/muting' includes options for static correction, dynamic correction, move starttime, muting, surgical muting, time cut, correct max. phase (selected), corr. max phase/wrap, correct picked phase, correct for 2 layers, correct 3Dtopography, and suppress multiples. The 'polarity' section has radio buttons for pos., neg., and first onset (selected). The 'zero cross.' section has a checked box. The 'correct time' section has radio buttons for no, auto (selected), and manual. The 'use mean' section has radio buttons for no and yes (selected). The 'lineparts' section has a radio button for lineparts. On the right, 'trace number' is set to 1, with buttons for 'original', 'filtered', and 'reference trace'. Below this are 'original spectrum' and 'filtered spectrum' plots. At the bottom, there are 'Filter parameter' fields: threshold (5000), nom.frequency [MH] (450), mean time [ns] (16.05503), and average(traces) (0). There are checkboxes for 'apply on example trace' and 'apply on original data'. A 'ControlPanel ProcessingLabel' is set to 0. At the bottom right, there are checkboxes for 'SequenceProc.' (checked) and 'close after processing', along with 'Start' and 'Close' buttons.

The **3D-data generation** is very simple. Use as type of interpolation use interpolation scheme for freely distributed 2D-lines and for example as X- and Y-rasterincrement 0.08 m and the double values for the interpolation. These are useful values because of the offsets between the dipoles of 8 cm within the Raptor array and the small interpolation allows a full coverage if the data are not acquired strictly along parallel lines. As sorting receiver coordinates must be entered.

The image shows a dialog box titled 'Generate 3D file from 2D-lines'. It has a '3D filename' field with 'file_3da'. The 'type of interpolation' is set to 'Use interpolation scheme for freely distributed 2D-lines'. There is a checked box for 'add k-y-scans by choosing the max. abs. value' and a checked box for 'apply mean filter in time direction'. The '3D coordinates' section has fields for XStart (498753.2), XEnd (498816.6), YStart (7178627.7), and YEnd (7178642.2). The 'X/Yrasterincrement' section has fields for XRasterincrement (0.08) and YRasterincrement (0.08). The 'X/Yinterpolation' section has fields for Xinterpolation (0.16) and Yinterpolation (0.16). The 'interpol weight' is set to 'linear weight'. The 'data scaling' is set to 1. The 'time range/sorting' section has fields for time end (50), timeincrement scale (1), and sorting (receiver coordinates). The '2D-files choice' section has a 'filepath' dropdown set to 'prodata', a 'filter' field set to '*.00i', and a 'load 2D-files' button. There is a 'min/max xy coord' field. At the bottom, there is a checked box for 'update traceheader coordinates' and 'Start' and 'Cancel' buttons. On the right side of the dialog, there is a text box showing '50. actual timeslice at: 9.5703' and 'depth: 0.23926 at v=0.05[m/s]'. Above the dialog box, there is a small image showing a radar data visualization with purple lines on a grayscale background.

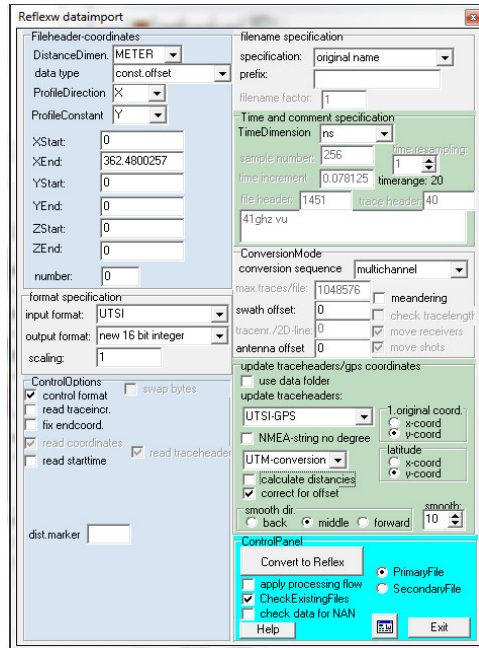
0.13.5. Utsi

The original Utsi data are saved as single 2D-lines.
The filename contains the channel (last character).

The Utsi system normally generates degree data stored on a file with the extension gps. An additional gpt file is generated which contains the tracenumbers of the gpr-file for the synchronization with the gps-file. An identical gps and gpt file is generated for all channels. As update traceheader type use **UTSI-GPS**. The relative positions of the antennas and the gps device must be defined manually within the correctoffset.txt file.

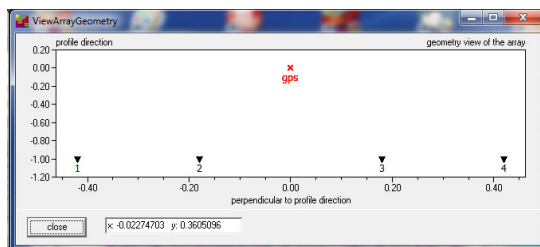
The option correct for offset allows to correct the offsets of the antenna in relation to the gps-device. Precondition is that the coordinates are in meter. Therefore a **utm-conversion** must be done if the gps-coordinates are given in degrees (set utm-conversion to UTM-conversion).

As mentioned before the antenna geometry must be entered manually using the ASCII-file correctoffset.txt.



Example file:

```
1 1.0 -0.42
2 1.0 -0.18
3 1.0 0.18
4 1.0 0.42
```



0.13.6. 3D-Radar

Reflexw only supports vol data from the 3D-Radar examiner program. The original 3D-Radar vol-data are saved within one single file containing all data of the antenna array. Reflexw automatically stores the files as individual 2D-lines with a filename extension indicating the channel number (number defines the naming value of the first channel).

The GPS-device is assumed to be located at 0/0 relative to the antenna array. If this is not correct the option **use correctoffset.txt** must be activated and the correctoffset.txt file must be adapted accordingly.

The gps data are stored within the navigation file with the extension nav. These are normally degree data. The options **1. original coord.** as well as **latitude** should be set to y-coord. The option correct for offset allows to correct the offsets of the antenna in relation to the gps-device. As the gps data are normally given as degree values the option UTM-conversion must be set in this case.

With activated option **"use correctoffset.txt"** the values within the existing correctoffset.txt file are used for the correction of the antenna array instead of the original data. If deactivated the original geometry is read in.

The original offsets are defined as: RelativeScanTime in combination with the mean velocity defined within the TriggerPosition block will be transformed into an offset in profile direction, lateral offset perpendicular to the profile direction is assumed to be positive right to the profile direction and vice versa. The file correctoffset.txt will be updated based on these geometries.

Example of four channels from 21 channels in total:

original nav-file

```
<TriggerPosition Lat="63.355818291253627" Lon="10.408339335089199"
Velocity="0.60743814706802368" Heading="201.91848754882812" DHeading="0.058531187474727631"
SourceType="Normal" IndexInPrimaryCube="1409" Time="2014-02-07T13:36:45.7270Z"
Elevation="187.61172485351562" />
```

```
<AntennaElementInfo TransverseOffset="-0.75" RelativeScanTime="0" TxRxDistanceX="-0.44"
TxRxDistanceY="0.074999999999999956" TxRxDistanceZ="0" TxElementNumber="0"
RxElementNumber="0" />
```

```
<AntennaElementInfo TransverseOffset="-0.675" RelativeScanTime="0" TxRxDistanceX="-0.44"
TxRxDistanceY="-0.0750000000000000067" TxRxDistanceZ="0" TxElementNumber="1"
RxElementNumber="0" />
```

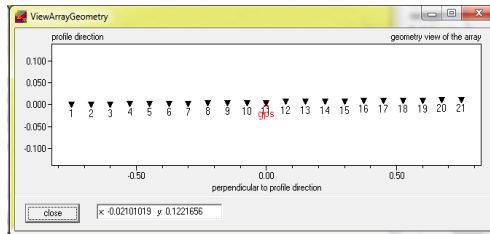


```
<AntennaElementInfo TransverseOffset="-0.6" RelativeScanTime="0" TxRxDistanceX="-0.44"
TxRxDistanceY="0.074999999999999956" TxRxDistanceZ="0" TxElementNumber="1"
RxElementNumber="1" />
```

```
<AntennaElementInfo TransverseOffset="-0.525" RelativeScanTime="0.002693631"
TxRxDistanceX="-0.44" TxRxDistanceY="-0.0750000000000000011" TxRxDistanceZ="0"
TxElementNumber="2" RxElementNumber="1" />
```

resulting correctoffset.txt file

```
1 -0.000000000000000E+0000 -7.500000000000000E-0001
2 -0.000000000000000E+0000 -6.750000000000000E-0001
3 -0.000000000000000E+0000 -6.000000000000000E-0001
4 -2.42163682900181E-0003 -5.250000000000000E-0001
```



0.14 general considerations

In this chapter the user will find answers to distinct questions which occur by the processing of data using ReflexW.

0.14.1 Which data types can be imported?

At the moment the program supports among others the following formats: SEG2, SEG2-RADAR, RADAN, RADAN_ANALOG, PULSEEKKO, MALA RD3, EMR, BGREMR, BGRDHU, BGRBHRU, UTSI, IDS, TRS2000, ABEM, BISON-2, OYO8BIT, ASCII-OYO8BIT, ASCIISYNTH, ASCII-3COLUMNS.

For further information please refer to chapter 1.5 (Import Menu).

There exist three different default data types in the Import Menu of ReflexW: const. offset, single shot and several shots.

The default data type 'const. offset' can be used for the import if the measurement is done along a line using a fixed offset between shot and receiver (e.g. Constant-Offset (CO) radar data).

If the data set is the result of a measurement with only one shot point but many receivers in a line, the use of the default data type 'single shot' data is recommended (e.g. Common-Mid-Point (CMP) radar data).

In case of data, which contain any kind of multiple shot / multiple receiver data, the default data type 'several shots' should be used (e.g. one data set combined of individual CMP data sets).

For further information please see below and refer to chapter 1.5 (Import Menu).

0.14.2 The difference between FileHeader and Traceheader

The fileheader contains all the information about the geometry of the whole file, e.g. it's start and end coordinates, distance dimension, trace increment.

Therefore, if one deals with equidistant data, the coordinates of any single trace are known in relation to the other traces of one file without defining the traceheader coordinates. Consequently, in most cases of radar data, no matter whether CO data or CMP data, the fileheader coordinates are sufficient to process the data (see chapter 7.5). PLEASE NOTE: The trace coordinates must not be mistaken for the traceheader coordinates!

The traceheader contains additional information concerning a single trace: e.g. shot position or receiver position in relation to the trace, which are needed in the case of multi shot data.

For further information please refer to chapter 1.9, especially to chapter 1.9.1 and chapter 1.9.6)

0.14.3 How is the FileHeader set or changed?

There exist different possibilities to set or change the fileheader:

First of all, the fileheader of every single file is set during the import. For further information please refer to chapter 1.5 (Import Menu).

To change and supplement the fileheader of one single file, please use the option Edit Fileheader in the File Menuitem of the 2D-Data-Analysis Module. For further information please refer chapter 1.9 (FileHeader Edit).

To change and supplement the fileheaders of several files belonging to one project, please use the option Edit several FileHeaders in the File Menuitem of the 2D-Data-Analysis Module. For further information please refer to chapter 1.10 (Edit several FileHeaders).

0.14.4 How is the TraceHeader set or changed?

In comparison to the fileheader coordinates which must be set in any case, the traceheader coordinates can be set if they are needed at all.

If the original file format provides them, it is possible to read them during the import. In that case the button 'read coordinates' has to be activated in the control options of the Import Menu (see chapter 1.5).

To change or set the traceheader coordinates of a single file after the import please use the option ShowTraceHeader in the option Edit Fileheader of the File Menuitem of the 2D-Data-Analysis Module. There exist three default possibilities to update the traceheader coordinates:

- They are read from an 'Ascii file' (*.DST), which contains the needed information for every trace.
- They are updated by using the fileheader coordinates.
- They are calculated using the 'fileheader' coordinates and the default type 'circle' if the receivers are orientated along a circle with the shot at the midpoint of the circle.

For further information please refer to chapter 1.9 (FileHeader Edit).

The traceheaders of several files belonging to one project can only be set or updated, respectively, simultaneously, in case of using the fileheader coordinates (option 'Update traceheaders' in the option Edit several FileHeaders in the File Menuitem of the 2D-Data-Analysis Module.). Otherwise, there exists no possibility to change or set the traceheader coordinates of several files at the same time. For further information please refer to chapter 1.10 (Edit several FileHeaders).

0.14.5 How are the PlotOptions set or changed?

Not only the fileheaders but also the plotoptions of every single file are set during the import. Therefore, it is recommended to set the desired plotoptions before the import of the data. For further information please refer to chapter 1.5.6 (ImportControlPanel) and chapter 1.7 (Plotoptions).

After having imported the files there are two possibilities to change the plotoptions:

To change the plotoptions of one single file please use the option PlotOptions of the option Edit Fileheader of the File Menuitem of the 2D-Data-Analysis Module.

For further information please refer chapter 1.9 (FileHeader Edit) and chapter 1.7 (Plotoptions).

To change the plotoptions of all or some, respectively, files of one project, please use the option PlotOptions of the option Edit several FileHeaders in the File Menuitem of the 2D-Data-Analysis Module. For further information please refer to chapter 1.10 (ProjectFile-Headers) and chapter 1.7 (Plotoptions).

0.14.6 Some special recommendations about geometry and preprocessing

Before starting the processing and interpretation of radar data the user of ReflexW must ensure, that the file header coordinates, trace coordinates (NOT to be mistaken for the traceheader coordinates) and the start time of the data are correct. Otherwise any information (e.g. material velocity or target position) extracted from the data is wrong!

0.14.6.1 Fileheader

Normally the fileheader coordinates set during the import are sufficient to process radar data, because in most cases one deals with equidistant data. Nevertheless, it can be necessary to enter some additional information in the fileheader: If the data file is e.g. a CMP, the position of the CMP point must be entered. For further information please refer to chapter 7.1.3 and chapter 1.9 (FileHeader Edit).

0.14.6.2 Coordinates

As already mentioned, the trace coordinates are set correctly during the import using the fileheader coordinates, if one deals with equidistant data. But there are some peculiarities:

If the data is measured by setting markers and not by using a survey wheel, the data must be preprocessed in the module data-analysis by means of the function markerinterpol to achieve correct trace coordinates (see chapter 1.11.6.1).

For some data like e.g. non equidistant data the traceheader coordinates (Hence the trace coordinates, too!) must be updated after the import. For further information please refer to chapter 7.1.3, chapter 7.1.4 and chapter 1.9.6 (Traceheader Edit).

0.14.6.3 Shot time

In the case of radar measurements it is not possible to measure exactly the moment at which the shot was generated, because one is dealing with a high frequency equipment. To ensure that no information is lost during the measurement, the data are often registered by starting the registration before the generation of a shot.

Therefore, the so-called shot time must be defined by a correction of the start time of the traces. This can be done in different manner dependent on the data:

In the case of single CMP data or single Moveout data, respectively, the shot time is defined by the direct air wave which intersects the time axes at the position of the CMP or the shot point, respectively. Often, the direct air wave has very small amplitudes in comparison to the other information in the data. In this case it is helpful to display the data using the plot option AGC-Gain (see chapter 1.7.5).

In default of distinct information the shot time for CO data is often set to that time, at which the first non zero amplitude is recorded, and the distance between transmitter and receiver is neglected.

For fixing the shot time of ZO data there is information needed about the time the radar system needs to switch from sending state to recording state.

For other data like non equidistant data, 3D data or VSP data no recipe for setting the shot time can be given.

In all cases the start time of the data has to be corrected in a way that afterwards the shot time is situated at $t = 0$ (see chapter 1.11.3.1). PLEASE NOTE: The correction of the start time does not automatically change the gain values, which were possibly set in the fileheader before! These values must be updated after setting the shot time according to the resulting time shift.

0.14.6.4 Peculiarities

Some data like MALA RD3 or PULSE-EKKO formatted data must be preprocessed in a special way using the module data- analysis: The first filter is either the so called subtract-mean(dewow) under the 1D-filter processing group (see chapter 1.11.1.8) or the subtract DC-shift filter (see chapter 1.11.1.9) which eliminates a possible time constant shift. The second filter consists of the application of a gainfunction (e.g. option manual gain(y), energy decay, AGC or gain function under the Gain processing group, chapter 1.11.2). The AGC (AutomaticGain Control) or the energy decay has not to be used as a filter function but can also be used only for the presentation of the data (see chapter 1.7.5).

0.14.7 Getting information about the velocity of the medium

There are various possibilities to get information about the velocity distribution in the investigated medium. Depending on the data, there are different ways to create 2D-velocity models, which can be used for further processing steps (see chapter 7.1.7.5).

0.14.7.1 Migration

If one deals with ZO- or CO-data with little offset, respectively, the energy, which was spread by any object, can be again concentrated at the place of the object by migration. This is done by e.g. summation along synthetic hyperbola, the shape of which is dependent on the velocity of the background. With the correct velocity the concentration of the energy is maximal. Therefore, if the background velocity is more or less homogeneous, the carrying-out of some migrations - by taking different velocities as a basis - will lead to the right background velocity. For further information please refer to chapter 1.11.9 (Migration)

0.14.7.2 ZO-data / CO-data

If the ZO- or CO-data with little offset, respectively, contains well defined hyperbola, e.g. originating from reflexions on pipes, synthetic hyperbola can be adapted to the data using the option velocity adaptation of the Analysis Menu-Item of the 2D-Data-Analysis Module. The shape of these synthetic hyperbola can be changed by e.g. varying the medium velocity or the radius of the pipes. It is also possible to determine a velocity by creating a straight line between two points. All adapted velocities can be stored on file with the extension hyp (e.g. test.hyp). When saving the current velocity adaptation, the following parameters are stored: velocity, distance in profile direction, x-weight. For further information please refer to chapter 1.12.1 (Velocity adaptation).

0.14.7.3 CMP-Measurements / Moveout-Measurements

If no hyperbola but reflectors are visible in radar ZO- or CO-data with little offset, respectively, CMP- or Moveout-measurements should be carried out to get information about the velocity distribution in the investigated medium. Every CMP- or Moveout data set can be processed within the module CMP-velocity-analysis to get a 1D-velocity-depth-model which can be stored. This can also be done for CMP sorted data because the module CMP-velocity-analysis can also be awaked from the option CMP-Processing of the Analysis Menuitem of the 2D-Data-Analysis Module. For further information please refer to chapter 2 (CMP velocity analysis) and chapter 1.12.4 (CMP-processing).

Another way to interpret CMP data is to adapt reflection hyperbola or straight lines to the data using the option velocity adaptation of the Analysis Menuitem of the 2D-Data-Analysis Module. Again, all adapted velocities can be stored on file

with the extension hyp (e.g. test.hyp). When saving the current velocity adaptation, the following parameters are stored: velocity, distance in profile direction, x-weight.

For further information please refer to chapter 1.12.1 (Velocity adaptation).

0.14.7.4 Using Coredata to get velocity information

A core data file contains the information of several cores associated with a 2D-line. This ASCII-file is either generated using real geological cores or may also automatically be created within the CMP 2D-model menu if the option create corefile is activated (see chapter 7.1.7.5 and chapter 2.3 (CMP 2D-model)). The core data file should be stored in the path ASCII in the current project directory and must have the extension cor (e.g. test.cor). For the format specification of a core data file please refer to chapter 1.4 (View Menu Item).

A core data file can be displayed as vertical bars onto a 2D-line by means of the option Core data/1D models in the View Menuitem of the 2D-Data-Analysis Module. If the core data do not contain the velocities of the individual core layers, all velocities are preset to the current adaptation velocity (option velocity adaptation of the Analysis Menuitem of the 2D-Data-Analysis Module). Within this option velocity adaptation you may interactively adapt the core bars to layer boundaries visible within the 2D data, if the speed button core is activated: Click on any layer bar which you want to adapt. The fill style of this layer bar changes. With the adaptation parameter v highlighted it is now possible to change the velocity of this distinct layer by pressing the key < or >, respectively.

Again, all adapted velocities can be stored on file with the extension hyp (e.g. test.hyp). When saving the current velocity adaptation, the following parameters are stored: velocity, distance in profile direction, x-weight.

For further information please refer to chapter 1.12.1 (Velocity adaptation).

There are some global settings influencing e.g. the display or saving of the core data files. For further information please refer to chapter 1.2.1 (Global settings). But there are two peculiarities which should be mentioned here: With the option save corefile in the option global settings of the Global Menuitem of the 2D-Data-Analysis Module activated the currently loaded corefile is automatically updated if the layer velocities are changed. With the option save VLA-file in the option global settings of the Global Menuitem of the 2D-Data-Analysis Module activated a VLA-ASCII velocity DataFile is automatically created when storing the core velocity adaptations on file with the extension hyp (e.g. test.hyp). This VLA-ASCII file may be used for the generation of a layershow. For further information about the layershow please refer to chapter 1.12.3 (LayerShow Menuitem).

With the global option **save vel,2dm-files** activated VEL-ASCII velocity datafiles for each core as well as a 2Dm-file (see CMP-velocity analysis) is automatically created when storing the core velocity adaptations which may be used for e.g. migration, timedepth-conversion, stacking,

0.14.7.5 Generating 2D-velocity models

2D-velocity models can be created either using the option velocity adaptation of the Analysis MenuItem of the 2D-Data-Analysis Module or by means of the option 2D-model in the CMP-velocity analysis module.

Using the suboption 2D in the option velocity adaptation of the Analysis MenuItem of the 2D-Data-Analysis Module it is possible to generate the 2-dimensional velocity distribution based on the current adapted or loaded, respectively, velocities. The interpolation is controlled by the parameter x-weight. For further information please refer to chapter 1.12.1 (Velocity adaptation).

The option 2D-model in the CMP-velocity analysis module allows the creation of a 2-dimensional velocity model on the basis of 1D-velocity-depth-profiles. For the creation of the distribution previously created 1D-velocity models are comprised to a 2D-model. For this the position within the profile line for the corresponding 1D-models has to be determined previously (see option model pos.). Between these 1D-models a linear interpolation is performed. The 1D-model with the largest model position is extrapolation to greater position values. The 1D-model with the smallest model position is extrapolation to smaller position values of the 2D-model.

PLEASE NOTE: Within the 2D-model file only the filenames of the chosen 1D-velocity models are stored. That is why changes in any of these 1D-models at a later stage have an influence, if the 2D-model file is used for any processing step.

For further information please refer to chapter 2.3 (CMP 2D-model).

0.14.8 Picking of onsets

The picking is done using the option Pick in the Analysis Menuitem of the 2D-Data-Analysis Module.

0.14.8.1 Preconditions for picking

If picking shall be done, the traceheader coordinates must be set to get the exact location of any pick. For further information please refer to chapter 1.12.2 (Pick Menuitem).

0.14.8.2 Setting of picks

The picks are set using one of the options manual picking, continuous pick, a semi-automatic picking using a phase follower and a special function called difference pick.

manual pick: With the option set activated a pick can be set with the left mouse button. The same applies to the manual removal or change of an existing pick after selecting the options remove or change, respectively.

continuous pick: With this option activated you can set, remove or change picks by moving the mouse cursor over the data set with continuously pressed left mouse button.

phase follower: The option allows the automatic assignment of picks to a selected phase. For this you first have to select the desired phase at an arbitrary position of the profile with the left mouse button.

difference pick: With this option activated the traveltime difference between two successive picks is automatically determined. In addition the thickness is calculated based on this traveltime difference and the entered velocity (see PlotOptions chap. 1.7). First you must enter the reference pick and then the investigation pick. The two values traveltime distance and calculated thickness are plotted right to the investigation pick using the symbol font. The remove button as well as the sort and control buttons are disabled. After having activated this option all picks are reset.

For further information please refer to chapter 1.12.2 (Pick Menuitem).

0.14.8.3 Saving of picks

The picks are saved using one of the following formats:

ReflexWin: new binary REFLEX pick format containing both the travel times and the amplitudes.

ReflexDOS: old binary REFLEX pick format.

ASCII-columns: convert picks to an ASCII file whereby every pick uses one line.

Ascii-tomography: also convert picks to an ASCII file whereby every pick uses one line. But the stored information differs from that stored using ASCII-columns.

For further information please refer to chapter 1.12.2.1 (Pick save MenuItem).

0.14.8.4 Saving of picks to create a layershow

If the option pick shall be used together with the option layershow in the Analysis MenuItem of the 2D-Data-Analysis Module, the picks have to be stored in the format ReflexWin. PLEASE NOTE: For using all options the layershow is providing, all the following information must be set before storing the picks:

To add a legend to the layershow, for every specific pick file a 'global code' has to be specified. As global code every combination of characters and numbers is allowed.

Additionally a 'velocity' has to be preset which is used for the conversion of the travel times in depths in the layershow if no other velocity information can be used (see chapter 7.1.8.6). You have to specify a mean velocity for the TOTAL overburden of the picked onsets (and NOT a layer velocity in between two sets of picked onsets).

Also, a 'layer number' must be specified, which serves to identify the individual picks within the layershow. Therefore, you should process together and save on a file only those arrivals that belong to one layer, if using the option pick together with the option layershow.

For further information please refer to chapter 1.12.2.1 (Pick save MenuItem).

0.14.8.5 Export of picks to create contour plots

To create contour plots, the picks must be exported using the format 'ASCII-columns' in one of the two following manners:

1. Export of current picks:

With the option 'export several existing picks into 1 ASCII-file' deactivated it is possible to save the current picks (no matter if just set or loaded from a pick file) into an ASCII-file, whereby every pick uses one line. Each line contains the following values: coordinate in profiledirection(10:3), coordinate in profileconstant(10:3), x-traceheadercoordinate (optional,10:xx), y-traceheadercoordinate (optional,10:xx), travel time(12:6), depth(10:4), amplitude(10:xx) where xx defines the number of decimal places given within the global settings menu (chap. 1.2.1). By default the travel times and also the plane-coordinates of the picks are written on a file. Additionally the travel time can be converted into depths on the basis of a specified constant velocity (option depth activated) and the current amplitude of each pick can be saved (option amplitudes activated). If no depths and/or amplitudes shall be extracted, these values are set to zero. Optionally the xy-coordinates of the receivers (specification xy-coordinates) stored in each trace header may be written out (see also FileHeader Edit). This allows you for example to write out non equidistant xy-coordinates or the xy-coordinates, if the profile is not orientated into one direction (x or y). The xy-coordinates are only saved on file if chosen. Optionally the acquisition times of each trace stored within the traceheader is

written out (option acquisition times activated). The parameter `n.pick` controls the pick number to be output: only every `n. pick` will be output. The file has the extension `PCK` and will be stored in the path `ASCII` in the current projectpath.

2. Export of several pick files:

With the option ,export several existing picks into 1 ASCII-file' activated you may export several existing pickfiles into 1 ASCII-file. Each line of the ASCII-file contains the following values: `traveltime(12:6)`, `depth(10:4)`, `amplitude(10:xx)`, `x-shot(10:xx)`, `y-shot(10:xx)`, `z-shot(10:xx)`, `x-receiver(10:xx)`, `y-receiver(10:xx)`, `z-receiver(10:xx)` where `xx` defines the number of decimal places given within the global settings menu (chap. 1.2.1). The `x`-,`y`-,`z`-shot and receiver coordinates have been taken from the traceheaders of the individual profiles when picking the onsets (see also trace header Edit menu - chap. 1.9.6). Depths and amplitudes are set to zero if the corresponding options `depth` and `amplitudes` are deactivated.

For further information please refer to chapter 1.12.2.1 (Pick save Menuitem).

0.14.8.6 Picking and time-depth conversion of several layer boundaries

To get information about the depth of arrivals/layer boundaries occurring in the radar data, the arrivals have first to be picked and then to be processed further on with the option `LayerShow` in the Analysis Menuitem of the 2D-Data-Analysis Module.

The picking has to be done as described in the preceding chapters. Afterwards the saved pick files can be loaded and displayed in the option `LayerShow`. 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.

For more information please refer to chapter 1.12.3 (`LayerShow` Menuitem).

For the time-depth conversion using the option `LayerShow` 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.

The second possibility is to use layer pick velocities or a VLA-ASCII velocity datafile (e.g. `test.vla`). In the latter case, the velocities for each layer may vary laterally and they are given as layer velocities in contrast to the mean pick velocities. Such a VLA-ASCII velocity datafile can either be generated using core files (see chapter 7.1.7.4) or using the option ,Create `LayerShow` velocities' in the `LayerShow` option (see below).

The option ,Create `LayerShow` velocities' (button ,create velocity' in the option `LayerShow`) can be used in two different ways to generate a VLA-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.

In the latter case there exist also two possibilities to proceed: The needed reference amplitude can be set manually or calculated based on a picked onset,

e.g. the reflexion from a metal plate. In the manual case, the reference amplitude must have been corrected for all necessary effects, i.e. a TRUE amplitude must be entered. If the reference amplitude is calculated based on a picked onset, their amplitudes need not to be corrected for all necessary effects. They will be corrected automatically if one of the following correction options are updated: ',correct gain curve', ',correct transmission losses' and ',correct geom. spreading'. These correction options influence beside the reference amplitude also the amplitudes of the current radar data during the calculation of the layer velocities using the amplitude calibration.

No matter how the layer velocities are calculated, they finally will be stored in a VLA-ASCII velocity datafile, which can be used to create a layershow.

PLEASE NOTE: Due to the huge number of possible effects which cannot be corrected for, the velocity determination based on the amplitudes must be very carefully interpreted!

For further information please refer to chapter 1.12.3.3 (Create LayerShow velocities) and chapter 1.12.3.4 (LayerShow amplitude calibration).

A new layer show is created using the ',create' button in the option LayerShow, which opens the create LayerShow window. This window allows to create a file of combined equidistant picks. The "Input-picks", which do not have to be equidistant, have to be saved on file before under the option PICK.

The creation is done by defining a filename for the layershow, its start and end position and an increment for the equidistant picks, which will be generated. Also, up to 200 pick files can be loaded. Finally, one has to choose, if the mean pick velocities, the layer pick velocities or a VLA-ASCII velocity datafile shall be used for the creation of the layershow.

PECULIARITIES:

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 the layer pick velocities or a VLA-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 current layer is done.

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

For further information please refer to chapter 1.12.3.1 (Create LayerShow).

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

For further information please refer to chapter 1.12.3.2 (Create LayerShow report).

1. 2D-Data-Analysis

The module **2D data-analysis** allows the complete processing of 2D-lines (B-Scans).

The possibilities:

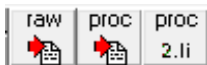
- easy import of data of many different formats (e.g. SEG-Y, SEG2, most of the EMR- and seismic systems (for example: GSSI, MALA RD3, PULSE-EKKO) and so on), integration of other nonstandard formats.
- conversion of single sections and of a profile sequence (automatic assembling and storing of the sections under one single datafile or automatic generation of datafiles for parallel and inline-sections).
- many different display and processing possibilities
- interactive processing of single files or by generating a batch-file (sequence processing)
- one- and multi-channel filters, editing, static correction, deconvolution, migration and much more.
- interactive velocity adaptation
- picking of onsets
- layer-show
- CMP-processing From this menu you may enter the main options.

Use the lower panel for a speed access to different options:

2.->1. button: Use this option if you want the secondary file to be the primary file (see also File MenuItem)

2<>1 button: Use this option if you want the secondary file to be the primary file and the primary to be the secondary

These options allow a fast access to the data.



The left button changes according to the last settings to caption raw: open REFLEX raw data (see also File MenuItem)

The middle button changes according to the last settings to caption proc: open REFLEX processed data (see also File MenuItem)

The right button changes according to the last settings to caption proc 2.li: open secondary REFLEX processed data (see also File MenuItem)

The caption proc 2.li: open secondary REFLEX processed data with filter (see also File MenuItem)

The caption raw 2.li: open secondary REFLEX raw data (see also File MenuItem)

The caption proc 2.li: open secondary REFLEX processed data (see also File MenuItem)

The caption proc 2.li: open secondary REFLEX processed data with filter (see also File MenuItem)

The caption is automatically updated when using one of the following options within the File MenuItem: open rohdata, open procddata, open procddata filter, import, open Second Line/rawdata or procddata or procddata filter

next button: allows to switch to the next primary file (ascending alphabetic order). The current filefilter for the primary file is taken into account. Example: the primary file "file001.01t" has been loaded

using the filenamefilter "01t". The next file must have the same extension "01t" and the next file therefore may be "file002.01t". If no filenamefilter has been specified the next file in ascending alphabetic order is chosen.

prev. button: allows to switch to the previous primary file (ascending alphabetic order) - see option next. Under special circumstances it may happen that the filefilter does not work correctly. If this is the case activate the option **double check filefilter** within the **global settings menu**. This may solve the problem.

- 1./2.: for the use with the next and prev. buttons: if activated the secondary profile will be correspondingly updated when using the next and prev. buttons. The specified filefilter for the secondary file will be used.
- 2.: for the use with the next and prev. buttons: if activated only the secondary profile will be updated when using the next and prev. buttons. The specified filefilter for the secondary file will be used. The primary profile remains unchanged.



print the current data files (see also Print Menu). Depending on the current settings either only the primary file or the primary and secondary file are printed based on the current splitting parameters. With the option showmarker activated (see PlotOption) the markers are printed in addition. The size of the markers are taken from the size defined within the symbol font (see FontSettings).



enter FileHeader Edit



enter PlotOptions



copy image to clipboard

jpg: copy image to file (jpg or other formats)

plotscale: enter multiplication factor for the color-amplitude assignment or for the wiggle size

velocity: enter a velocity for the transformation of the time scale to a depth scale for the depth axis and for the layer image.

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



the options allows an automatic scrolling of the file in x-direction if a zoom has been activated (scroll to the left, stop scrolling, scroll to the right). The spin button located beneath the autoscroll options defines a delay factor in order to slow down the scrolling. Value 0 no delay - value 500 max. delay. If the option file has been activated the autoscroll allows an automatic passing through all files of a project (switch to the next (right arrow) or previous file (left arrow)). Again the delay value controls how fast the switching to the next file is done.



allows to easily switch between pointmode and wigglemode



allows to directly change the display split - ver. split, hor. split, overlay and no split

Pecularity for overlay: if a second file has been loaded and overlay has been activated, the display of the second file may be enabled/disabled by simply activate/deactivate overlay. A possible zoom will be kept if the option keep/kept has been used.

If more than 2 profiles have been loaded it is possible to splite the windows vertically horizontally, vertically and horizontally and manually



If manually has been chosen click on one item (1..4) within the CheckListBox and use the mouse with pressed mouse button in order to exchange two items (this item with any other one). The option ver., hor. and split define the type of display splitting.



replot current line with current zoom parameters



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



enable magnifying glass function



enable manual zoom - With the option ZOOM an arbitrary area of the data set can be selected and plotted in full screen size. If the screen is split in order to show two data sets, this option can be applied to each data set, but not across the separating line of the two data sets. With split screen the ZOOM option automatically acts on both data sets.

The area to be enlarged, a rectangle, has to lie within a data set. Pressing the left mouse button you determine a corner of this rectangle and by moving the mouse with pressed button the desired area is opened.

The zoom range may be changed step by step using the small + or - buttons on the right side. Clicking on the + button increases the zoom by 10 %, clicking on the - button decreases the zoom by 10 %, this means the x-y-range to be shown will be larger.

The current zoom may be stored using the small "s" button and may be restored using the small "g" button.



If activated it keeps automatically actual zoom settings when performing a filter action or loading a new profile using next or prev. button as well as when using the 2<->1 or 2-> buttons.



enters the velocity adaptation Menuitem

pick button:

enters the Pick Menuitem



enter the LayerShow Menuitem

CMP button:

enters the CMP-processing

x-dist. button: allows to measure the distance between 2 points in horizontal (x) direction. The program waits for the input of two data points for generating a straight line. Pressing the left mouse button determines the first data point and by moving the mouse with pressed button the line for measuring the x-distance is shown. Leaving the mouse button holds the current line and shows the distance in horizontal direction.

Core: the option is identical to the option View/Core data/1Dmodels (Chap. 1.4, more details are given there). Activate this option if you want to display core data (1D-depth distributions) as vertical bars onto the primary line and onto the layershow depth profile. With the option **auto** activated the corefile with the same filename like the primary file will be loaded automatically. The postposition text entered within the pick panel (2D-dataanalysis) for the automatic load of the picks is used in additon. This allows you to specify a different filename for the automatic load. The option has the same effect as the option autom.load within the global settings menu.

3-comp. button: enters the 3-component analysis menu (chap. 1.12.8).

**GPS button:**

check parity: <input type="radio"/> yes <input type="radio"/> no	DataBits: <input type="radio"/> 7 Bits <input type="radio"/> 8 Bits	Baudrate: <input type="radio"/> 2400 <input type="radio"/> 9600 <input type="radio"/> 4800 <input type="radio"/> 14400	com port: <input type="radio"/> 1 <input type="radio"/> 2 <input type="radio"/> 3 <input type="radio"/> 4	GPS trace at: <input type="radio"/> start <input type="radio"/> end <input type="radio"/> middle	corr. GPS pos: <input type="checkbox"/> activate synchronization	delay: <input type="text" value="100"/>	<input checked="" type="checkbox"/> UTM to Degree <input type="checkbox"/> UTM 32n	scroll type: <input type="radio"/> stepwise <input type="radio"/> smooth 2 <input type="radio"/> smooth 3 <input type="radio"/> smooth 4
--	---	--	---	--	--	---	--	--

The option allows to control the display of the current profile by GPS-coordinates coming through the serial port. The option opens a new panel where you can enter the necessary parameters for the serial port.

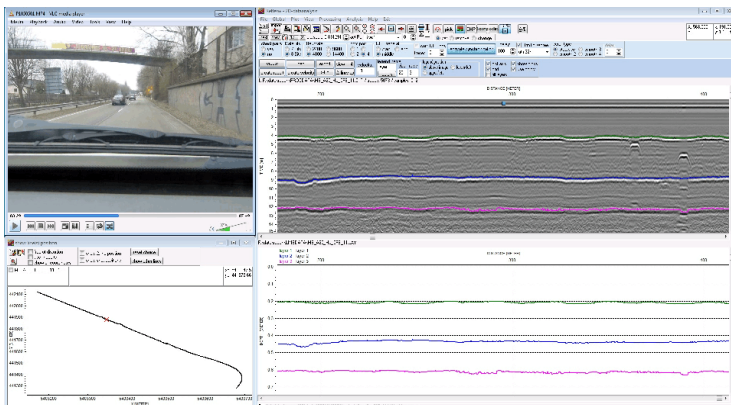
The option might be useful if you want to display a camera video together with the radargram. The synchronization will be done by the gps-coordinates stored within the Reflexw traceheaders and the gps-coordinates coming through the serial port. The device GeoAudio+ from Intuitive Circuits for example allows such a synchronization. The device with integrated GPS receiver converts GPS position, heading, speed, altitude, date and time into a continuous audio stream that can be recorded by standard camcorders. The converted audio data is automatically synchronized with the camera's video image.

The device may be used afterwards in order to decode the audio stream into GPS-coordinates and outputs the coordinates as a NMEA string using the serial port.

The option **activate synchronization** starts the synchronization with the NMEA strings. Each time a NMEA string has been recognized at the entered com port the programs skips to the position given within the NMEA coordinates. The delay is necessary for the address of the com-port. The radio group box **GPS trace at** allows you to define where the trace with the found GPS coordinates is situated either at the start, at the middle or at the end. The position will be marked by a blue rectangle. Because of the delay of the GPS-system the GPS mark may not be correct. Therefore it is possible to define a second mark for the **corrected GPS position**. The correction range must be given in traces. The position will be marked by a blue circle. The NMEA string must have the following form:

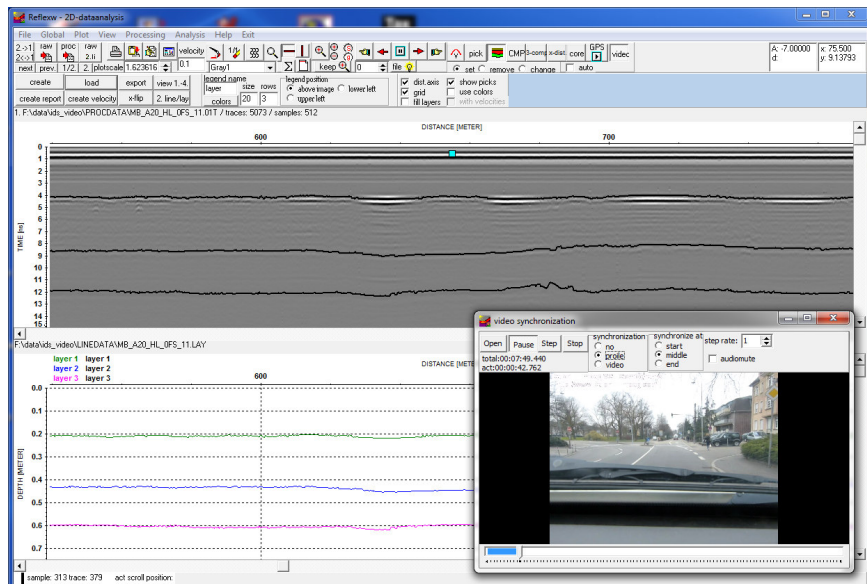
\$GPGGA,095656,3757.3888,N,02348.9613,E,1,09,1.3,914.8,M,34.4,M,0000*4A
The coordinates are given in degrees. If the profile has been converted to UTM-coordinates the option **UTM to Degree** must be activated and the corresponding UTM zone must be chosen.

The **scroll type** box allows you to define a stepwise or a smooth skipping. If smooth has been activated you can choose a delay value in order to adapt the scrolling. Three smooth steps are available.



video-button:

The option allows to control the display of the current profile by a video and vice versa.



Precondition is a txt file of the following format:

Each line contains

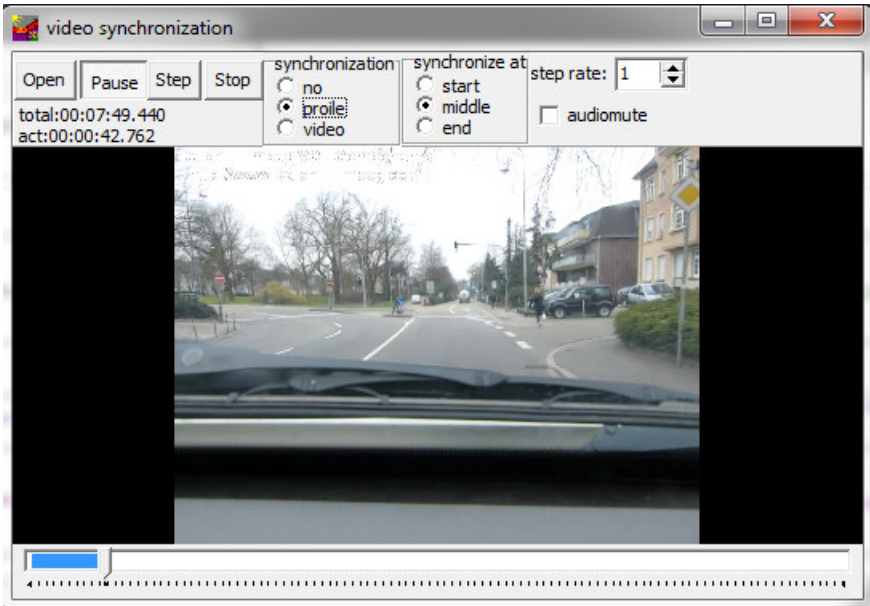
- the video time in hour:minute:second
- a Scan_ID identical to the tracenummer which will not be interpreted
- a Swatz_ID which will not be interpreted
- the distance along the profile

Example in which the gpr profile starts at the video time 10 sec:

```
00:00:1.000 0 1 0.00
00:00:2.000 0 1 0.00
00:00:3.000 0 1 0.00
00:00:4.000 0 1 0.00
00:00:5.000 0 1 0.00
00:00:6.000 0 1 0.00
00:00:7.000 0 1 0.00
00:00:8.000 0 1 0.00
00:00:9.000 0 1 0.00
00:00:10.000 0 1 0.00
00:00:11.000 100 1 5.00
00:00:12.000 200 1 10.00
00:00:13.000 300 1 15.00
00:00:14.000 400 1 20.00
00:00:15.000 500 1 25.00
00:00:16.000 600 1 30.00
00:00:17.000 700 1 35.00
```

The SRS Sync Video application from IDS already creates such a synchronization file which partly may be used here. First such a file must be created for each profile and second the filename must be renamed in such a way that it fits the original data filename.

The option opens a new panel where you can enter the necessary parameters for the synchronization.



Open allows to open a video file, e.g. a mp4 file. The video synchronization text file with the same filename but with the extension txt must exist. Otherwise an error message appears.

The synchronization panel allows to synchronize the gpr profile with the video (profile activated) or vice versa (video activated).

The synchronize at panel allows to specify the location of the actual video within the gpr profile (start, middle or end).

The step rate allows to set a rate for the synchronization.

Activate the option audiomute in order to mute the audio.

The option Pause pauses the video and the synchronization. Step allows to step one video screen forward. Stop stops the video and the synchronization.

The trackbar at the bottom allows to switch to any part of the video.

Within the **Main Menu Bar** you may choose the following items:

File Menuitem: From this menu you may open any primary or secondary file, import data from different formats, export the data to different formats, copy the current images to the clipboard or print out the data.

Processing Menuitem: From this item you may choose the different filter and edit possibilities.

View Menuitem: This item controls some display options.

Global Menuitem: Enter some global settings like fonts, printer,...

Plot Menuitem: Within this icon you may activate all the plot settings.

Analysis Menuitem: This menu allows you to activate analysis procedures like picking or velocity adaptation.

key shortcuts:

There exist some key shortcuts for a fast access:

F2 - opens the plotoptions menu

F3 - deactivates show second line and screen split

F4 - increases the actual pick code by 1

F5 - decreases the actual pick code by 1

F6 - dataanalysis - copy to clipboard

F7 - save actual zoom settings

F8 - get actual zoom settings

“<” and “>” keys: the plotscale value can be changed using these keys. The step size of 20 % corresponds to the step size of the plotscale up/down arrows. The keys cannot be used if the interactive velocity analysis is activated as in this case the keys are used for changing the adaptation parameters.

General viewing options:

The working panel at the bottom contains a small black rectangle which shows the actual zoom. Next the actual sample number as well as the actual trace number is shown when moving the mouse. If a zoom is active the actual scroll position of the primary file (mid position of the screen) will also be displayed.

The panels at the upper right corner contain the actual x and y-values of the mouse position as well as the actual amplitude (option amplitude under view active) and the depth (if a depth axis is shown) and the xyz-receiver traceheader coordinates (options xyz-traceheader coordinates under view active).

1.1 File MenuItem

From this menu you may open any primary or secondary file, import data from different formats, export the data to different formats, copy the current images to the clipboard or print out the data.

ProjectDir: Selection of a new project. After selecting this option the directories included in the current project directory are listed. From these you can choose the project you want to work with.

Open: Use this option to load an existing REFLEX-file into the primary data window. You may load the data from the rawdata directory (the directory containing the files which are not preprocessed) or from the procd data directory. If you are choosing procd data filter a filter for the file extension must be entered in order to make the choice easier. In addition you may load a REFLEX-file interactively from the profile plan (see also interactive choice).

Open SecondLine: Use this option to load an existing REFLEX-file into the secondary data window. You may load the data from the rawdata directory (the directory containing the files which are not preprocessed) or from the procd data directory. If you are choosing procd data filter a filter for the file extension must be entered in order to make the choice easier. The procd data filefilters can be chosen independently for both the primary and the secondary file. The secondary data are plotted into the second image depending on the current screen split (horizontal split, vertical split or no split - see also Plotsuboptions).

The secondary data file cannot be processed. The file is only used for a comparison with the primary file. When a processing is done for the primary file the secondary file is automatically overwritten with the current processed file.

Open 1.-4. Line: Use this option to load up to 4 different REFLEX-files into different windows. The data windows may be splitted vertically (vertical split activated) or horizontally (horizontal split activated) or both (options vertical split and horizontal split deactivated). The chosen files are automatically sorted in alphabetic ascending order. The option 2<->1 allows you to change the order. The first line becomes the last one, the second the first one and so on.

The print, zoom and scroll options are available for all 4 lines whereas picking and velocity adaptations can only be done within the standard 2 window configuration. When a processing step is performed always the datafile of the first window is the base for the processing and the datafile of the second window is replaced by the new processed datafile.

ChangeSecondToPrimary: Use this option if you want the secondary file to be the primary file. For example use this option if you want to use a filter sequence on the primary file - e.g. 1.filter: bandpass, 2.filter: run.average.

- apply the bandpass filter on the primary file - in the secondary window the bandpass filtered data are displayed
- choose option ChangeSecondToPrimary
- apply the run.average filter on the new primary file - in the secondary window the completely filtered data are displayed (bandpass filter and run.average).

Print: Use this option if you want to print out the data. Depending on the current settings either only the primary file or the primary and secondary file are printed

based on the current splitting parameters (see also Print Menu). With the option **showmarker** activated (see PlotOptions) the markers are printed in addition. The size of the markers are taken from the size defined within the symbol font (see FontSettings). With the option **Pick** activated the current picks are printed in addition. The size of the picks are taken from the size defined within the symbol font.

Import: import data of different formats into the REFLEX-internal format (see also Import Menu).

Export: allows the conversion of REFLEX-formatted data into different export formats (e.g. SEG Y) - see also Export Menu (chap. 1.6).

Edit FileHeader: This option allows to display and change the parameters stored in the global file header and the individual trace headers. These parameters are for example sample and trace increment, presetting of coordinates etc (see also FileHeader Edit).

Edit several FileHeaders: opens the Edit several fileheaders menu (chap. 1.10). Within this menu you may change the fileheader coordinates and the start time (see also fileheader time specification.) of a choosable number of lines.

Edit TraceHeader: opens the trace header menu (chap. 1.9.6).

Processing flow: displays the processing flow of the current primary file. The option **save to pfl** allows to save the processing flow on file with the extension pfl. Such a file can be used within the sequence processing. The option **reset** resets the processing flow of the current file (old processing flow will be lost). With the option **several files** activated the processing flows of a number of choosable files will be reset. The option **export to ASCII** allows to write the processing flow on an ASCII-file (filename will be queried and the file will be stored with extension PFL.txt under the project directory). With the option **several files** activated the processing flows of a number of choosable files are written out.

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

Image1ToClipboard: copy the primary window into the clipboard.

Image2ToClipboard: copy the secondary window into the clipboard.

Exit: leave the module 2D-dataanalysis.

1.1.1 interactive choice

Within this menu the profiles may be interactively selected from a profile map. A preselection of the profiles to be selected is done by the option **Filepath** and **Filefilter**.

Filepath specifies the path for the files to be shown on the profile map. **Filefilter** allows to enter a filter for the files to be shown. e.g. filepath is set to `procddata` and filefilter is set to `*.02t` - only the files with the extension `02t` are shown on the profile map and can be selected.

Additionally you must select the plane where profile to be selected are located. You have the choice between **XY-plane**, **XZ-plane** and **YZ-plane**. Only those profiles whose profile direction and profile constant span this plane, are projected.

With the option **show profile** activated the currently chosen profile is shown within a separate window whereby a fast display of the different profiles is possible. The plotting depends on the choice of the profile direction (see below). For example a profile in x-direction is plotted with a horizontal distance axis wherey a profile in y-direction will be plotted with a vertical distance axis. The size of the distance axis is automatically determined from the profile length within the profile map but the plotting window may be rescaled and moved.

With the option **free of distortion** activated the x- and y-axis are equally scaled for display.

With the option **use traceh.coord.** activated the traceheader coordinates (receiver) are used for the display of the profileline.

Activating the option **show filenames** displays the the filenames within the middle of the profile.

With the option **show markers** activated the markers of each profile are shown in addition. The precondition is that the markers are stored within the traceheaders.

With the option **show currecnt picks** activated the current picks are displayed in addition.

Activating the option **show all lines** reads the coordinates from all profiles located within the given plane and shows them as straight lines and within the input listbox on the left hand side in addition.

Activating the option **ResetChoice** also reads the coordinates from all profiles located within the given plane and shows them as straight lines. In addition already chosen profiles are removed from the output listbox.

Activating the option **show picks** allows you to display the xy-projections of the wanted picks. After having activated the option you are asked for the pick files to be loaded. The option is useful for a direct control of the locations of the picks within the xy-plane. Picks belonging to the same code are connected if the option interpolation is activated. If the option **use code** within the 2D-dataanalysis or 3D-datainterpretation is activated the **layershow-colors** are used for the display of the picks. For example the option might be used for the location control of picked pipes or rebars.

With the option **interpolation** activated the xy-projections of the picks belonging to one element (given by the pick code) are connected (see option show picks).

The profiles may be selected or deselected either within the input listbox by clicking on the checkbox or within the profile plane by clicking on the line. For a selection the **set** button must be activated and for a deselection the **remove** button must be activated. A deselection may also be done within the output listbox by activating any selected file and press the delete key.

The interactive selection within the profile plane is additionally controlled by the **"direction"** buttons. Only profiles can be selected or deselected, whose direction complies with the active direction. Already selected profiles are marked red, all the others black. For the selection of the profiles always that - not yet selected - profile is taken that is next to the mouse cursor. If there exist more than 1 profile with identical coordinates within the plane, then after selecting or deselecting a listbox appears with all profile names having these coordinates. Also the beginning of the profiles is not marked as usually by the symbol "X" but by a number, signifying the number of profiles with identical coordinates within the plane.

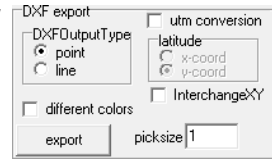
To be considered: The max. number of profiles to be selected depends on the REFLEX module from which the interactive choice has been called. When calling from the 2D-dataanalysis the max. number is restricted to 4 (see also option Open 1.-4. Line, chap. 1.1). When calling from the 3D-datainterpretation the max. number is restricted to 25. With the option show profile only 1 profile may be selected.

It is possible to select files from different project directories and different search paths because the outputlist box contains the full filenames.

The option **Btm** allows to load a background bitmap, e.g. a photo of the measure plane. The actual clipboard may also be directly pasted using the key Ctrl V. Precondition is that the profiles of the profile map are already loaded and therefore coordinate axis are present. After having activated the option Btm you are asked for the wanted bitmap. The bitmap is plotted in addition and a menu opens where the size of the background bitmap may be entered. By default the bitmap is plotted filling out the complete coordinate range under the assumption that the coordinate origin is located on the lower left corner (flipped y-axis). Within the menu you may enter a different coordinate range of the bitmap and to deactivate the bitmap flipping in y-direction (option **FlipYAxis**). After having defined the correct coordinate range of the bitmap you may use the option **close** in order to close this menu.

If a tfw file is present the coordinates are taken from this tfw world file (see also Geotiff export). Thereby a georeferenced bitmap may be loaded as a background. Deactivating the option Btm removes the background bitmap. The option Btm is also used when printing out the profile plan.

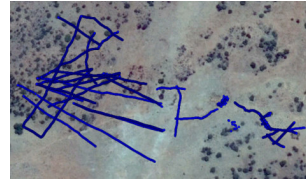
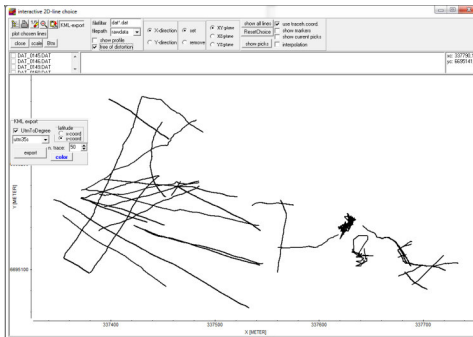
DXF export: activating this option opens a new window for exporting the profile lines into the DXF format. The option DXFOutputType controls if the lines will be displayed as points or lines. An optional **utm conversion** is possible. With the option **InterchangeXY** activated the xy-coordinates will be interchanged for the export. This might be usefo for example if the latitude has been stored on the x-coorindate and no utm conversion shall be done. If the option different colors is activated the different lines will have different colors. The actual picks will also be exported as circles. The option picksize controls the size of the picks. The **layer** group code within the dxf-file will be set to the pickcode which allows an easy discrimination of the picks.



The option **scale** allows an manual scaling of the axis and the coordinate range to be shown. If activated the coordinate range window is shown and with the option manual scaling activated a manual scaling is enabled.

The option **plot chosen lines** allows to plot the actually chosen profiles without closing the interactive choice menu.

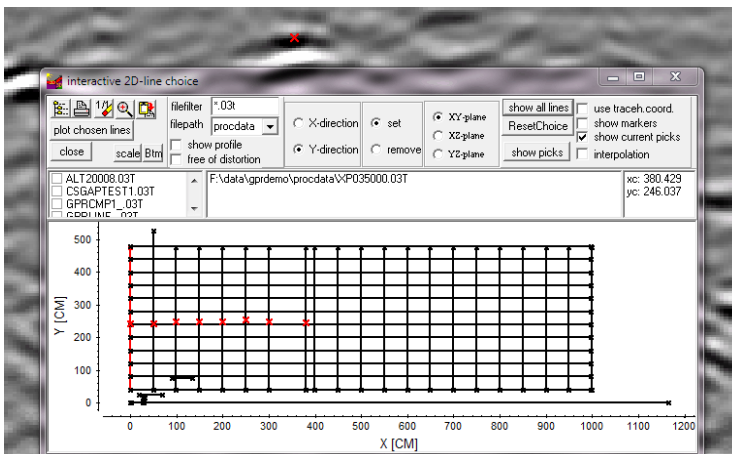
KML export: allows to export the profile line coordinates to a kml file which may be loaded afterwards by e.g. GoogleEarth. The color may be defined as well as the number of icons (n.trace). The option UTMTToDegree allows to convert UTM-coordinates to degree values based on a given utm zone.



The option **show current picks** allows to show the current picks. In combination with the options next or prev. you may easily follow a distinct reflector or diffractor (e.g. a pipe) from one 2D-line to the next and display the current position and the positions of the previous locations within the interactive choice map. Use for example the following sequence:

- load the first file using file/procdata filter and enter the name filter (e.g. *.03t)
- activate the option view/interactive choice
- within the interactive choice menu activate the option show current picks
- activate the pick option within the dataanalysis menu and pick your wanted arrival(s) (the current cursor position as well as the actual pick(s) are displayed within the interactive menu)
- click on next and choose the pick(s) of the same event (the interactive choice menu can be used for following e.g. a line diffractor)

The option **close** closes the interactive choice and the selected profiles are plotted. You may also leave the menu by the key escape. In this case all changes done



within the interactive choice window are lost and the program returns to the calling module.

1.2 Global MenuItem

printersetup: enters the printersetup menu.

info: enters the info windows about REFLEXW.

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

cursor: specify the wanted cursor.

global settings: this menu allows to specify some global settings parameters (see also global settings)

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

1.2.1 Global settings

this menu allows to specify some global settings parameters:

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

ScrollSize: enter the fracture part of the image to be scrolled by one scroll step. For example ScrollSize 1/10 means that scrolling is done with a step rate of 10 % of the total size of the image. $\frac{1}{2}$ means a step rate of 50 %. The new scrollsize is only available after a new zoom range has been entered. The current setting is stored within the ini-file when leaving the program. After having changed the scroll size you must reset a possible zooming using the option reset and afterwards reenale the zoom range in order to activate the new scrollsize.

synchronize scrolling: if activated the scrolling of the primary and secondary file within the 2D-dataanalysis is synchronized. The current setting is stored within the ini-file when leaving the program.

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

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

Double check filefilter: if activated the filefilter for the next/prev. buttons within the 2D-dataanalysis will be double checked.

White background for free datarange: if activated the background color given within the plotoptions is used for printing or bitmap export for a free area range. A free area range might be present if the manual scaling option is used. If deactivated the 0 amplitude value color is used.

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

Bitmap size[kb]: this parameter controls the max.size of a bitmap created within REFLEXW. The default value is 16318 kb. The max. allowed bitmap size may vary from computer to computer. Therefore you may decrease this parameter if any problems occur e.g. during printing within the 3D-datainterpretation. The program automatically restricts the bitmap to this max. value and generates different bitmap parts if necessary.

GPS Map API key: The GPS Map uses an API key for most map services except OpenLayers. The API key for which you have to register within the map service like Google Maps or Bing platform must be defined here. Reflexw uses the dynamic maps which is only loaded once when entering the map service. Therefore the fees will normally very low, if any. The key will be stored within the Reflexw.Ini file.

NMEA String: with \$G*GGA the standard NMEA string like GNGGA will be analysed. With \$GNLLQ activated a special GNLLQ string will be analysed which normally contains Gauss Krueger coordinates.

BackgroundColor: choose the background color for the wiggle display.

Read exist.par-files for direct import: if activated the program looks for an existing REFLEX-parameter file when a file is directly imported using double click within explorer.

MarkerGroupBox: enter some parameters for the display of the comment and distance markers:

construction change marker: enter the text for the so called construction change marker (see also edit comment markers, chap. 1.12.6). A vertical bar is plotted in addition to the comment marker text if the comment marker string is the same like this text. In addition this text is used within the Create LayerShow velocities menu (chap. 1.12.3.3 - option use construction change).

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

update from GPS ASCII-file if activated a gps marker will be placed at all positions defined within the Ascii-file used for updating the traceheader coordinates (see chap. 1.9.6).

Add. Filename for comment.fil: if activated the filename comment.fil for the comment markers will be replaced by the actual profile filename (without extension) plus _comment.fil, e.g. file0001_comment.fil. Thereby each profile file will receive an own comment marker file.

MarkerType: you can choose between a rectangle at the top of the trace or a dashed line over the complete trace for the display of the distance markers.

pick parameters: With this groupbox you may enter some parameters controlling the picking (see also chap. 1.12.2).

With the parameters **PickSymbols** you can choose between three different symbols for the picks (*, - and line). If line is activated the picks are plotted using an interpolated line between the picks. The option is inactive if a single pick is set, changed or removed. In this case the symbol '-' will be used.

The parameter **display every n.pick** controls that only every n. pick will be plotted. This does not affect the settings of the picks themselves. Enter a number larger than 1 if the phasefollower is too slow or if the picks are not displayed when scrolling fast.

With the parameter **pickcorrect traces** you may enter the number (one sided) of traces for the pick correction option corr.max/min.time (see also Pick MenuItem).

The parameter **corr.xy-coord bin(%)** controls the size of the crossing bin when you are using the option control xy-coord within the pick panel (see chap. 1.2.2). The parameter is in percentage of the current traceincrement. Increase this parameter if the option control xy-coord does not generate any picks.

With the option **show pickdifferences** activated the traveltimes differences and thicknesses are displayed when using the difference pick.

With the option **keep distance** activated the x-position between the two picks are plotted in addition when using the difference pick. With this option not activated the x-position of the second pick is automatically moved to the x-position of the first one.

The option **decimal places** controls the number of decimal places for the ASCII-export of the picks.

The option **places** controls the number of places for the ASCII-export of the picks.

plot on 2.line: if activated the picks are also displayed on the 2.line whenever you press the plot or reset button. For example this option may serve as a controlling possibility of the picking if you have acquired the same data with two different frequencies. With the options **show cursor in 2. window** (under 2D-dataanalysis/view) activated the actual picks will be continuously displayed within the secondary profile.

core parameters: Within this groupbox you may enter some parameters controlling the coredata (see also option Core data/1D models under View MenuItem). The option **bar size** controls the size in pixels of the bars of the individual cores. With the option **show border** activated a black border rectangle is plotted in addition to the colored bar. With the option **show label** activated the labels of the cores are shown in addition. The label corresponds to the individual corename without CORE. With the option **save corefile** activated the current corefile is updated when storing the core velocity adaptations (see also velocity adaptation). With the option **save VLA-file** activated a VLA-ASCII velocity datafile (see Create LayerShow velocities) is automatically created when storing the core velocity adaptations which may be used for the generation of a layershow. With the option **save vel,2dm-files** activated VEL-ASCII velocity datafiles for each core as well as a 2Dm-file (see CMP-velocity analysis) is automatically created when storing the core velocity adaptations which may be used for e.g. migration, timedepth-conversion, stacking, With the option **autom. load** activated the corefile with the same filename like the primary file will be loaded automatically. The postposition text entered within the pick panel (2D-dataanalysis) for the automatic load of the picks is used in addition.

This allows you to specify a different filename for the automatic load. The option has the same effect as the option **auto** to be found below the speedbutton core within the 2D-dataanalysis main panel.

With the option **use xy-traceh.coord.** activated the traceheadercoordinates are used for determining the position of the core. The position for secondary profiles is only correct if the traceheader coordinates are identical to those of the primary profile.

Within the radio box **quality factor** you may define how the quality factors within the core files will be used:

ignore: the quality factors will be ignored

show: if activated a second bar is shown indicating a quality factor stored within the core datafile. The following colors are used: 1 - white, 2-yellow, 3-red.

use f.colors: if activated the colors for the layer bars will not be assigned to the layer number but to the quality factor using the layershow colors assignment.

CMP-velocity parameters: Within this groupbox you may enter some parameters controlling some functions within the CMP velocity analysis (see also chap. 2).

The option **display scale** allows to set the relation of the model display in percent of the total window width. By default a value of 25 % is set.

The option **NMO scale** allows to enter the size in percent of the nmo-correction window in comparison to the data window.

The option **analyse scale** allows to enter the size in percent of the analysis window in comparison to the data window. Enter a smaller value for the semblance and a larger value for the constant velocity analysis.

The option **curves size** allows to enter the number of discrete points of the calculated traveltimes curves within the CMP velocity analysis. The larger this value the larger the needed time to compute the traveltimes. Therefore you should search for the best compromise between a good resolution of the traveltimes curves and a fast refresh of the display. By default a value of 20 is set.

The option **color** allows you to enter the color for the calculated traveltimes curves within the CMP velocity analysis.

The option **pen size** allows you to enter the pensize in pixels for the calculated traveltimes curves within the CMP velocity analysis. The possible valuerange is from 1 (thin) to 5 (5 - very thick).

The option **repaint images** within the global setting menu controls the display refresh. If activated the images will always be repainted when a model change has been done. Deactivate this option if a canvas error occurs. Activate the option if the images need a refresh.

LayerShow parameters: Within this groupbox you may enter some parameters for the layershow like the plot colors (option **color**) and pen-thicknesses (options **thick** and **thick line**) of the individual layer boundaries (option **number**). The option **thick line** allows to enter the overall line thickness for those layer boundaries for which the option **thick** is activated.

Model and traveltimemeanalysis parameters: Within this groupbox you may enter parameters for the modelling and traveltimes analysis module. The option **def. distance dimension** defines the used dimension in x-distance (either METER or FOOT). Depending on this setting the velocity dimension is either m/s (m/ns) or ft/s (ft/ns).

use shot for discr.: if deactivated the shot numbers are not used for the discrimination of the traveltimelines within the modelling menu

use code for discr.: if activated the codes stored with the picks are used in addition for the discrimination of the traveltimelines within the modelling or the traveltime analysis menu. Activate this option for example if different pickfiles for one shotpoint (e.g. refractions and reflections) are loaded or calculated. Then the code stored with the picks is used in addition to discriminate the traveltimelines (the picks with different codes are not combined using a line).

max.diff. for traveltimelines: enter a distance value for the discrimination of the traveltimes which will be combined with a line within the modelling menu. The option might be useful for example if two traveltime branches for one shot have been picked but the picks are missing in the neighbourhood of the shot. Enter a value smaller than the distance of this gap in order to prevent that this gap will also be combined by a straight line.

After changing any of these three parameters the traveltimes must be loaded again in order to take effect.

1.3 Plot MenuItem

enter these options either from the main menu or from the main tool bar.

PlotOptions

Plot: replots current line with current zoom parameters

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

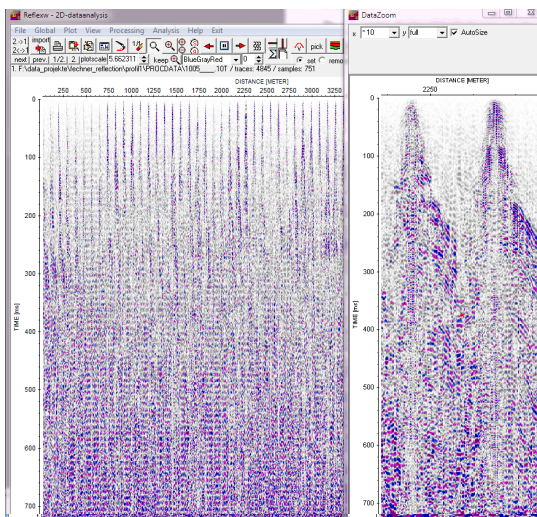
Autoscroll-Right: autoscroll line to right

Autoscroll-Left: autoscroll line to left

Autoscroll-Stop: stop autoscroll

ManualZoom: enable manual zoom. Click on the uppermost left corner of the area to be zoomed up and drag the mouse to the wanted lowermost right corner with the left mouse button pressed. The two additional speed buttons in the tool bar enable a steplike increase or decrease of the zoom factor.

MagnifyingGlass: enable magnifying glass function. A freely definable data part is continuously magnified within a window when moving the mouse. The zoom factors in x- and y-direction are independently definable. With the option **AutoSize** activated the vertical size of the zoom window will be automatically adapted to the size of the time axis of the original profile. The zoomed display will be completely renewed with high resolution when moving the mouse.



1.4 View MenuItem

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

Amplitudes: activate this option if you want the current amplitudes to be displayed in the status panel on the top (A:) when moving the mouse. If a secondary profile will be loaded and overlaid the amplitudes of the secondary profile will be displayed below (A2:).

PickPanel: activate this option if you want the pick panel always to be visible even if the pick option is not chosen.

VelocityAdaptationPanel: activate this option if you want the velocity adaptation panel always to be visible even if the velocity adaptation option is not chosen.

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

act sample informations: if active a window opens near the actual cursor position showing the actual trace and sample number, the amplitude, the distance and the time.

WiggleWindow: This option allows the additional plotting of the current trace or of the trace spectrum or of a second profile into freely movable and scalable window. The y-axis corresponds to that of the primary profile. After activating the option you may choose between **current trace**, **ampl. spectrum**, **phase spectr.** or **2. line**. After activating **current trace** the current trace (only visible part) is plotted in the wiggle-mode using the current Plotsettings. Activating the option **Tracenormalize** enables a normalization to the max. value of the current trace.

After activating the option **ampl. spectrum** the frequency amplitude spectrum of the current trace is shown. Only the visible part of the current trace (e.g. if zoomed) will be taken into account for the spectrum calculation. The option **frequ.scale** allows to rescale the display of the **ampl.spectrum**. The frequency range to be displayed will be decreased by the entered **frequ.scale** value.

After activating the option **phase spectr.** the frequency phase spectrum of the current trace is shown. Only the visible part of the current trace (e.g. if zoomed) will be taken into account for the spectrum calculation. The calculated phase is restricted to the range between 0 and 2 pi.

Activating the option **2. line** allows the plotting of a second profile in the wiggle-mode. Open the wanted line using the option **open**. All traces of the profiles are shown.

With the option **view sec.trace** activated the corresponding trace from the secondary profile will be plotted in addition using a gray color.

With the option **normalize** activated the display will be normalized to the max. value within the total time window. If deactivated the scale can be changed using the up and down arrows.

With the option **AutoSize** activated the vertical size of the wiggle window will be automatically adapted to the size of the time axis of the original profile.

With the option **dock** activated the wiggle window will be placed at the right side of the 2D-dataanalysis window if enough space is available. If not enough space is

available the size of the data window will be decreased and the wiggle window will be placed right to the data window. In this case the option dock becomes the color aqua.

The current parameters time (frequency) and amplitude are shown if the mouse cursor is inside the wiggle window.

TraceHeaderDistances: activate this option if you want the current traceheader distance to be displayed in the status panel (dist.:) .

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

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

The dimension of the thickness and of the location is identical to the current distance dimension of the profile. The dimension of the velocity is m/ns or m/s respectively.

A block ends when the program finds the identifier CORE or at the end of the file.

example:

```
CORE 1 - dist.dim: METER, vel.dim: m/s
14.00 0.00 0.00
1 7.887100 1 470.553 470.553
2 11.804501 1 576.020 677.474
3 15.052200 1 717.993 1059.677
CORE 2 - dist.dim: METER, vel.dim: m/s
24.00 0.00 0.00
1 7.887100 1 470.553 470.553
2 11.219000 1 573.388 677.474
CORE 3 - dist.dim: METER, vel.dim: m/s
34.00 0.00 0.00
1 7.887100 1 453.631 453.631
2 11.219000 1 562.827 677.474
3 15.637701 1 713.371 1059.677
```

The **x-location** corresponds to the distance. The **y-location** will not be used within the 2D-dataanalysis by default. With the option **use xy-traceh.coord.** within the global settings menu activated the traceheadercoordinates are used for determining the position of the core. Then the smallest difference between the xy-location of the core and the xy-traceheader coordinates of the actual profile is searched and the core will be plotted at this position.

The **z-location** will be considered if the plotoption DepthAxis has been activated. With the plotoption elevation activated in addition the z-locations are assumed to be elevations, if deactivated they are given in depths. The given z-location will be transformed into 2-way traveltimes using the velocity defined for the depth axis if the data are timebased data (normally the case).

The max. number of different layers within one coreblock is 100 (identical to the max. number of layers within the layershow). The colors for the individual layerbars are identical to the colors within the LayerShow MenuItem. With the option **show quality** within the global settings activated a second layerbar is shown indicating the quality factor with the following color assignment: 1 - white, 2-yellow, 3-red. Within the global settings menu you may also change the width of the layerbars and activate or deactivate some checking parameters.

These core data are also the basis for a coredata based velocity adaptation (see also velocity adaptation - suboption **core**).

A core datafile may also automatically be created within the CMP 2D-model menu if the option **create corefile** is activated.

With the option **save VLA-file** within the global settings menu activated a VLA-ASCII velocity datafile (see menu Create LayerShow velocities or Create LayerShow - suboption velocity choice / velocity file) is automatically created when storing the core velocity adaptations which may be used for the generation of a layershow.

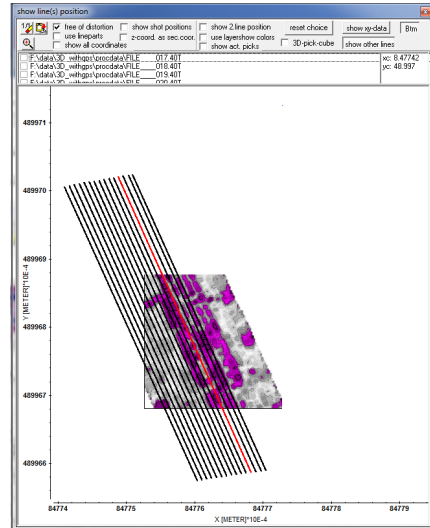
interactive choice: opens the interactive choice window without going through the option file/open/interactive choice.

profile line (trace header coord.): with this option activated the profile location based on the traceheader coordinates (see also trace header Edit, chap. 1.9.6) is shown in an additional window (any curvature of the line coordinates is displayed). When moving the mouse cursor within the data window the current xy-position of the mouse cursor is also shown. With the option **free of distortion** activated the x- and y-axis are equally scaled for display. The option **show other lines** allows to show the xy-locations of other profiles. Activating the option **show filenames** displays the the filenames within the middle of each profile. The option **reset choice** resets this display of the other profiles. The option **show 2. line position** displays the xy-position of the secondary file using the same distance coordinate. With the option **z-coord. as sec.coor.** activated the secondary coordinate is assumed to be the z-coordinate, if deactivated the y-coordinate is used. With the option **show shot positions** activated the shot positions stored within the traceheaders are displayed in addition in green (blue for 2.line) color. With the option **use lineparts** activated the coordinates lines will be displayed under consideration of the line parts (if the ensemble number changes the coordinates will not be combined using a line). With the option **show all coordinates** activated all coordinates will be plotted. Deactivating the option means that the coordinate line will be resampled based on the max. number of 1000 for the display. With the option **view single points** activated the profile line will not displayed as a line but as single points. Activating the option might be useful in order to check for irregularities within the traceheader coordiantes, e.g. looking for jumps due to an inaccurate gps signal.

Zooming and copy to clipboard possibilities are included. Activating the option **use layershow colors** uses the predefined layershow colors for the display of the lines. The profile lines will be automatically updated when using the next or prev. buttons. The option **show xy-data** allows to display points with xy-coordinates taken from an ASCII-file. The first column must contain the x-coordinate and the second column the y-coordinate. The size and color are defined by the symbol font. The option **show act. picks** allows to view the actual picks. With the option **3D-pick-cube** the picks are displayed within the 3D-cube in addition.

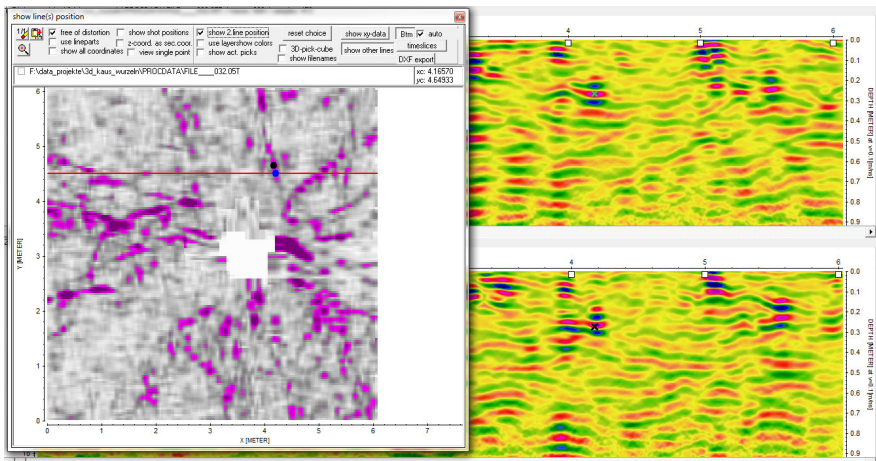
The option **Btm** allows to load a background bitmap, e.g. a photo of the measure plane. The actual clipboard may also be directly pasted using the key Ctrl V. Precondition is that the profiles of the profile map are already loaded and therefore coordinate axis are present. After having activated the option Btm you are asked for the wanted bitmap. The bitmap is plotted in addition and a menu opens where the size of the background bitmap may be entered. By default the bitmap is plotted filling out the complete coordinate range under the assumption that the coordinate origin is located on the lower left corner (flipped y-axis). Within the menu you may enter a different coordinate range of the bitmap and to deactivate the bitmap flipping in y-direction (option **FlipYAxis**). After having defined the correct coordinate range of the bitmap you may use the option **close** in order to close this menu.

If a twf file is present the coordinates are taken from this twf world file (see also Geotiff export). Thereby a georeferenced bitmap may be loaded as a background. Deactivating the option Btm removes the background bitmap. The option Btm is also used when printing out the profile plan.



The options **Btm/auto** together with **timeslices** allow an automatic display of bitmaps., e.g. timeslice bitmaps.

The option **auto** together with **Btm** allows to load a series of bitmaps, e.g. timeslices which will be updated when moving the mouse cursor within the primary profile. The option may be of help when picking linear elements which are clearly visible within the timeslices. Activating the option **invisible** makes the control panel invisible and no axis will be displayed. This might be useful if a small window



is required. Double clicking onto the interactive plan makes the control panel visible again.

The option **timeslices** allows to create the wanted bitmaps. The parameters within the GenerateTimeslices window are the same as for the option generate single Timeslices within the 3D-datainterpretation.

All profiles chosen with activated option show other lines within the show line position menu will be automatically taken over within the timeslice menu. Some new parameters which are needed for the new bitmap function have been included:

interpolate for pure x-and y-lines: if activated the choosen 2D-lines are assumed to be purely or nearly purely distributed along the x- and y-axis. In this case the interpolation will be done within 2 steps. The first interpolation is only done in x-direction for the profiles orientated into the y-direction. The second interpolation is only done in y-direction for the profiles orientated into the x-direction. Afterwards both datasets will be added.

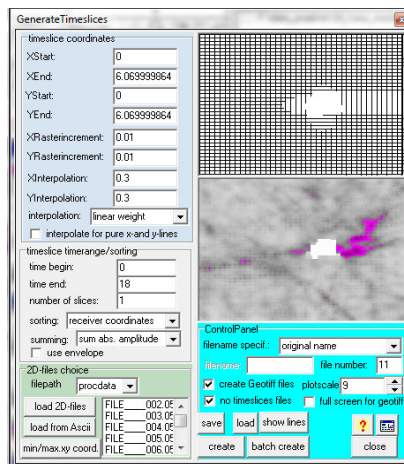
use envelope: the envelope will be used for the generation of the timeslices

create Geotiff files: generates a tiff file together with a tfw world file containing the coordinates.

no timeslices files: no timeslice file will be generated. This is only useful if create Geotiff files has been activated

full screen for geotiff: the normal size of the tiff file corresponds to the size of the picture above. If full screen for geotiff has been activated the full size of the GenerateTimeslices window will be used for the tiff file. This allows a higher resolution.

plotscale: enter the plotscale for the display of the timeslice. If a Geotiff file will be generated the plotscale must be correctly entered here because the bitmap cannot be changed afterwards.



When moving the mouse within the primary profile within the 2D-dataanalysis the actual time of the mouse position is used in connection of the filenames of the bitmap in order to determine the corresponding bitmap timeslice.

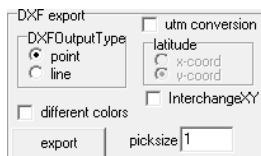
Example for automatic naming:

settings: time begin: 100 msec, max. time for calculation of timeslices: 500, number: 0 - bitmap timeslice receives the name TP100000.TIF (tif bitmap) and TP100000.TFW (tfw file containing the corner coordinates) and is saved within in the directory ASCII.

Example for manual naming:

settings: time begin: 5 nsec, max.time: 50 nsec, FileName: Test - time slice receives the name Test0500.TIF (tif bitmap) and Test0500.TFW (tfw file containing the corner coordinates) and is stored in the directory Procdta.

DXF export: activating this option opens a new window for exporting the profile lines into the DXF format. The option DXFOutputType controls if the lines will be displayed as points or lines. An optional **utm conversion** is possible. With the option **InterchangeXY** activated the xy-coordinates will be interchanged for the export. This might be usefull for example if the latitude has been stored on the x-coord and no utm conversion shall be done. If the option different colors is activated the different lines will have different colors. The actual picks will also be exported as circles. The option picksize controls the size of the picks. The **layer** group code within the dxf-file will be set to the pickcode which allows an easy discrimination of the picks.



Gps map: allows to view the profile lines within a GPS Map service like OpenLayers, Google, Bing, Azure, TomTom, Here and MapBox.. The default service is OpenLayer. The option **map service** allows to choose any of the other ones.

If you want to switch between the different Map services you must have different API keys. Only the OpenLayer service does not need a valid key. If you switch for example to Bing, you should first enter the valid Bing key within the global settings menu if not already done- option **GPS Map API key**. This key will be stored within the Reflexw.Ini file.



The option **UTMToDegree** allows to convert UTM-coordinates to degree values based on a given utm zone. The option latitude defines where the latitude coordinates have been stored within the traceheaders (x- or y-position).

The **color** and the **width** of the lines may be adapted. The display of the profile line may be disabled by setting width to 0. This might be useful for example if the pick display (see below) is hidden by the line. The display of the lines might be quite timeconsuming. Therefore the program automatically restricts the number of points per line to be displayed to 1000 points or if more than 10 lines (max.points 250) or

more than 50 (max.points 100) lines are shown. Independently from these settings you also may restrict the number of points manually using the option **max.points**. With the option show center activated a center point of the line will be displayed in blue.

Activating the option **show filenames** displays the start and end point and the filename. Activating the option **show line position** allows to continuously show the actual mouse position by a red circle when dragging over the line. Activating the option **show other lines** allows to show the xy-locations of other profiles. With the option **show shot positions** activated the shot positions stored within the traceheaders are displayed in addition in green (blue for 2.line) color. **Zooming** and **copy to clipboard** possibilities are included. Activating the option **use layershow colors** uses the predefined layershow colors for the display of the lines. The option **1/1** resets and centers the display.

The display will also be automatically centered when using the zoom-option or when scrolling through the zoomed profile. The actual profile center will be displayed by a red circle.

Activating the option **use bitmap** generates an internal bitmap copy of the gps map which is useful under some circumstances for viewing the picks or for viewing the actual line position. For example the actual line position will not be displayed for a very large file. In this case use bitmap must be activated which allows the view. If the option use bitmap is activated you should not flip to any other profile within the 2D-dataanalysis but first you deactivate the option and then choose the new profile and afterwards you may activate the option again. Otherwise the gps map display does not work any more and you must close the window.

The actual **picks** may be displayed (option **show** activated) using a red-green-blue color scheme for the icons with red corresponding to the smallest traveltime values and blue to the largest traveltime values. The parameter **max. picks** controls the max. picks to be displayed. If the actual number of picks exceeds this number only every n. pick will be displayed. If the option use bitmap is activated the color scheme for the pick traveltimes will be displayed at the left border. If 3D-picks have been loaded the option **use depths f.colours** controls if you want to use the colours for the depth assignment instead for the codes of the 3D-picks. The option has no meaning for only profile based picks.

To be considered: if for any reasons the gps map does not work correctly any more you may try to close the gps map window and open it again but in some cases the Reflexw program must be completely restarted.



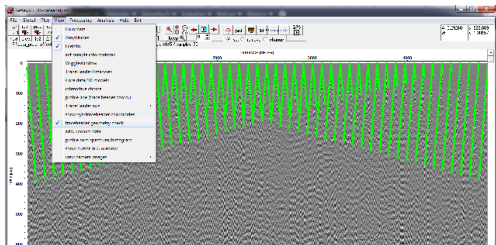
TraceHeader axis: if activated the xy-receiver traceheadercoordinates are displayed along the distance axis in addition. The option is only available for the primary profile and with deactivated plotoptions Rotate90degree and FlipXAxis. The number font is used. The number of places and the number of decimal places may be entered within the global settings menu within the pick parameters panel. The option transp.comm.marker within the global settings menu controls if the text will be transparent or if a white box will be underlied.

The suboptions at the top, at the bottom and at min.distance define the positions of the traceheader coordinates. With at the top activated the traceheader coordinates are displayed at the distance labels within the upper range of the profile below the distance axis using a 90 degree rotated font. With at the bottom activated the coordinates are displayed within the lower range of the profile again using a 90 degree rotated font. With at min.distance activated only the traceheader coordinate for the min. coordinate will be displayed above this min. coordinate. If enough space above the min. coordinate is available the x- and y-traceheader coordinates will be separately plotted, otherwise they will be combined separated by the "/" character (according to the options at the top and at the bottom). With at the top activated also the stored collect times (option time collect) may be displayed at the top of the profile. Otherwise and at the bottom or at min. distance the traceheader xy-coordinates will be displayed.

show xyz-traceheader coordinates: shows the actual x-, y- and z-traceheader coordinates (xc, yc and zc) within a new listbox on the upper right corner.

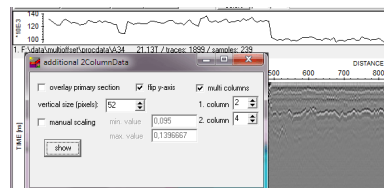
Traceheader geometry check:

allows to view the shot-receiver geometry. The actual velocity within the plotoptions is used in order to define the travelttime between shot and receiver used for the display. The option is useful in order to check the geometry of a dataset consisting of several shots, e.g. a seismic reflection profile.



profile line within second window: the option allows to show the actual profile of the primary profile within the secondary and vice versa. The option only makes sense if the primary window contains a timeslice and the secondary one a 2D-profile.

add. 2column data: option allows to display any 2 column ASCII-data (e.g. the topography) along the profile (distance) axis.




After having activated the option show within the Add2ColumnData menu you are asked for the ASCII-file with each line containing two values - the distance and the parameter value. These parameter data may contain for example any informations (e.g. topography, moisture,) along the profile. The max. number of datavalues is restricted to 32768.

After having chosen the wanted ASCII-file a control menu opens. The option **overlay primary section** control whether the data are plotted within a separate image at the top of the profile (option deactivated) or within the current primary section (option activated). The size of the vertical axis is given by the option **vertical size** (pixels) The horizontal size is connected to the size of the profile distance axis. The min. and max. parameter value of the chosen ASCII-file is automatically determined. Activating the option **manual scaling** allows to restrict the range to be displayed. With the option **flip y-axis** activated the value-axis starts at the bottom.

With the option **multi columns** activated allows to use multi columns text data. The option **1.column** defines the column position of the distance data and the **2.column** defines the position of the data column to be displayed. This allows you to use data with more than 2 columns, e.g. picks stored using the ASCII-columns format. Example: enter 2 for 1. Column if the distances are stored within the second column and 4 for 2. Column if the data values have been stored within the 4. column. The data are plotted as a polygonal line. The line color as well as line thickness is taken from the symbol font. The option is not available for the printing.

show cursor in 2. window: shows the actual cursor position within the secondary profiles using the settings of the symbol font.

profile sum spectrum/histogram: activating the option opens a new window which shows the **amplitude histogram** or the **sum trace** of the primary profile (upper panel) as well as the **sum spectrum** over all traces of the profile (lower panel).

enable manual zoom  - if activated an arbitrary area of the data set can be selected. The sum spectrum and the histogram will be calculated for this range. The range will be displayed within the table.

Use actual zoom - if activated and manual zoom deactivated the actual zoom settings within the 2D-dataanalysis are used for the calculation of the sum spectrum and the histogram.

The number of traces and/or samples to be summed may also be entered within the trace/sample table. The larger/smaller arrows on the right of the table allow to increase or decrease the min./max. traces and samples. Using this option also allows to scroll through the whole profile by setting min./max. traces to the same value.

Double clicking on the histogram/sum trace or the spectrum copies the corresponding chart to the clipboard for a succeeding import into any document.

The following options are only available for the **spectrum calculation**:



copy to clipboard

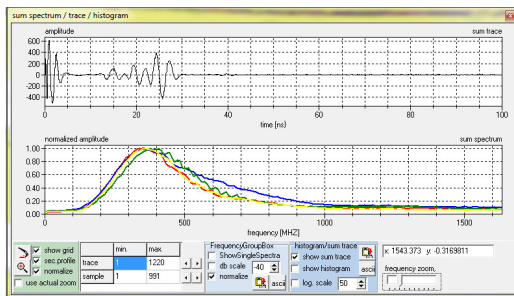
ASCII: allows to copy the sum spectrum to an ASCII-file (csv extension).

Trackbar which allows to change the max. frequency

ShowSingleSpectra - if activated the spectra of the single traces are plotted in addition to the sum spectrum.

db scale: allows a decibel decay scale for the frequency spectrum. The max. db decay may be entered manually

sec.profile - if activated the spectra of the secondary profiles will be shown in addition (blue 2. line, green 3. line, yellow 4. line). With the option **normalize** activated the spectra of the secondary lines are normalized to the spectrum of the primary profile.



With the option **normalize** within the FrequencyGroupBox activated the frequency spectrum will be normalized to 1.

The following options are only available for the **histogram**:



copy to clipboard

ascii within histogram/sum trace: allows to save the actual histogram/sum trace on an ASCII-file (csv extension).

show sum trace - if activated the sum trace will be shown

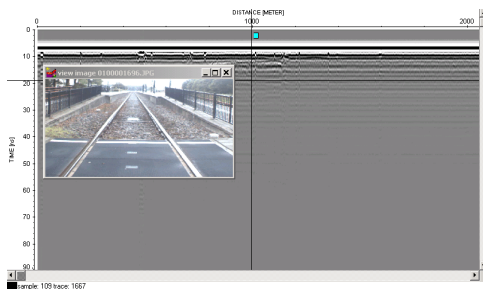
show histogram - if activated the histogram distribution is displayed. It is possible to select the number of subdivisions for the histogram. The default value is 50. Activating the option **log.scale** enables a logarithmic scale for the amplitudes.

view camera images: if activated camera images are shown based on the actual cursor positions within a freely moveable and scalable window. Two options are available: marker and tracenumber.

If **tracenumber** has been chosen the filenames of the camera image are directly connected to the tracenumbers of the profile. After having activated the option the camera image files must be selected (multiple file choice using the ctrl or shift key). The filenames of the camera images must have the following form:

The last 8 characters of the filename specify the trace number position within the profile, e.g.:

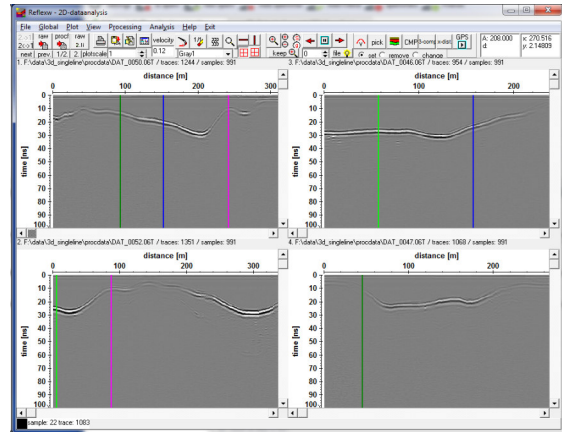
0100003396.jpg - the string 00003396 is used in order to determine the tracenumber position of the camera image.



If **marker** has been selected one

file within the camera directory must be chosen in order to determine the path where the camera image files which have been stored within the comment markers in a step before are present. All comment markers of the profile which text has an extension are assumed to represent a camera filename stored within the chosen path (see option edit comment markers/create based on filenames, chap. 1.12.6). The advantage using this procedure in order to view the camera images is that processing steps which change the trace number may be applied on the data. While moving through the profile the camera image next to the actual cursor position will be plotted. The position of the camera image will be marked as a blue rectangle at the top of the profile.

crossingpoints: if activated it allows to display the location of the crossing points of the actually loaded lines. The layershow colors are used in order to assign the related crossing points. The parameter **corr.xy-coord bin(%)** under global settings (chap. 1.2.1) controls the size of the crossing bin together with the traceincrement. Increase this parameter if no crossing points are found although the lines are really crossing. The picture on the right shows 4 crossing profiles (profile location is given within small lower picture on the right).



1.5 Import Menu

Use this option to import non REFLEX formatted data. The maximum number of traces per file is limited to 256000. At the moment the program supports the following formats:

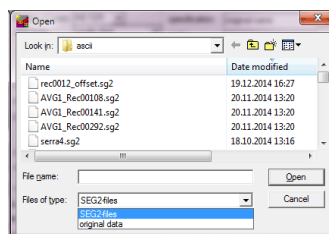
SEG2, SEG2, SEG2-RADAR, RADAN, PULSEEKKO, MALA RD3, MALA RD6, MALA RD7, IDS, UTSI, IMPULSERADAR, INFRASENSE, RADSYS, GEOSCAN32, BGREMR, BGREMRHELI, DMTKR, BGRDHU, BGRBHRU, TRS2000, ABEM, DECOD, CSP, NWF, NICOLET, YOKOGAWA, FREE 8BIT, FREE 16BIT, FREE 32BIT, FREE 32BIT FP, FREE 64BIT FP BIT, HP-network, UWB-MEODAT, ASCII-1COLUM, ASCII-1COLUM-ALPS, ASCII-3COLUMNS, ASCII-GORILLA, ASCII-MATRIX, ASCII-3D-KOEHLER, BITMAP, FD-total wavefield, 3D-Radar, ASCII-Proceq-PD80, KGS, JSF, PS3-Sonne, Geoscanners_Gsf, VIY, HCD

The program performs a so called plausibility test, i.e. various values (e.g. the number of samples per trace) are checked for reasonable values. If an "incorrect" value is recognized (e.g. number of samples smaller than 0 or larger than 16000), the conversion is terminated with the corresponding message.

The default directory for the non REFLEX data is ASCII under the current project directory (default import directory) but it is also possible to import the data from any other directory or drive. Example for storing the externally formatted data under the default import directory: Standard directory name is C:\REFLEX, inside C:\REFLEX the project directory DEMO1 is created (C:\REFLEX\DEMO1), inside this the default import directory ASCII (C:\REFLEX\DEMO1\ASCII). Into the directory C:\REFLEX\DEMO1\ASCII the desired externally formatted data might be copied.

In any case REFLEXW stores the converted data under the path ROHDATA under the current project directory. REFLEXW DataView stores the converted data under that path the non REFLEX data are placed.

For most import formats an automatic **filefilter** will be used (see picture on the right). If another fileextension for the original data is given you may select original data within files of type instead.



It is possible to **automatically import** a datafile from the formats RADAN, PULSEEKKO, MALA RD3 and IDS by double click on such a file provided the corresponding datatype (DZT, DT1, RD3 or DT respectively) is connected to ReflexW. Example: you want to automatically import a MALA RD3 file - you must connect REFLEXW with files with the extension RD3. After double click on such a MALA RD3-file, e.g. within the explorer, REFLEXW is started and this file is automatically imported and displayed. The following import options are always valid: filename is set to original name, read traceincrement is always activated. The imported datafile is stored under the path rohdata under the projectpath. The projectpath is automatically determined from the path where the original file is located. Therefore it is not possible to load a datafile from CD. If the last directory within this path is named ASCII (standard import directory - see above), the directory ASCII is neglected for the determination of the projectpath, otherwise the projectpath is equal to the path where the original file is located. Example1: the

original file is located under d:\reflex\demo\ascii - the converted file is stored under d:\reflex\demo\rohdata. Example2: the original file is located under d:\reflex\demo\name1 - the converted file is stored under d:\reflex\demo\name1\rohdata.

If the option apply processing flow (see also below) is activated the processing flow named importprocessingflow.pfl if existing under the actual project directory will be automatically applied during the import. This processingflow file can also be copied to the actual project directory if already created for an earlier project and if the data shall be processed accordingly.

The settings of the following options are stored within the Reflexw.ini file and therefore will also be used for this automatic import:

apply processing flow

read starttime

single file header (RADAN Data)

swap bytes (SEG Y Data)

min./max. marker (RADAN data)

Before beginning the conversion you have to specify several settings. These are essentially coordinate settings and name specifications. These specifications also enter the naming of the converted data, if this shall be done automatically or not. REFLEX stores the data using a 16 bit integer format or 32 bit floating point format. If your original data have a larger dynamic range than the output format you must enter a scaling value (option scaling). All data are multiplied with this value before conversion. With 16 bit outputformat specified the resulting values larger than 16 bit are clipped during the conversion.

Note: during conversion all 8 bit-data are internally multiplied by the factor 16 in order to be able to use all default parameters of the program (e.g. plotting parameters) without any essential changing.

Most of the available informations are read from the data headers. There is one peculiarity concerning the geometry. The program REFLEXW is based on a 3-dimensional geometry. That is why the coordinates must be entered manually and those parameters are normally not read from the file header. Peculiarities for the following formats:

PULSEKKO-Format: With the option read coordinates activated the starting and ending coordinates in profile direction are read from the header. With the option read traceincr. activated the trace increment is read from the header and the ending coordinate is calculated based on the manually given starting coordinate, the trace increment and the number of traces. With no option activated the entered manual parameters are valid. With the option **16 bit data** activated the original data are stored as 16 bit integer values. With the option deactivated the original data are assumed to be 32 bit floating point values. A quite big scaling value is recommended (e.g. 10000) in this case.

MALA RD3 and RD7-Format: The datavalues of the rd7-data normally exhibit quite large values. Therefore it is recommended to rescale the data during the import using a small scaling values of e.g. 0.00002. This ensures that the resulting datavalues fall into the "normal" 16 bit integer data range. In any case the output format should be set to new 32 bit floating point in order to maintain the full resolution.

With the option read coordinates activated the ending coordinate in profile direction is read from the header (the starting coordinate is set to 0). With the option read trace increment activated the trace increment is read from the header and the ending coordinate is calculated based on the given starting coordinate, the trace increment and the number of traces. With no option activated the entered manual parameters are valid.

The program supports three different marker files (same filename but different extension as the rd3- or rd7-file):

mrk-file: ASCII file with each line containing the tracenummer of the marker. The markers are written out as distance markers.

mrk-file: ASCII file with one header line and then each line containing the tracenummer of the marker. The markers are written out as distance markers.

mrkj-file: ASCII file without linefeed with specification of the marker positions about the flag "trace:". The markers are written out as distance markers.

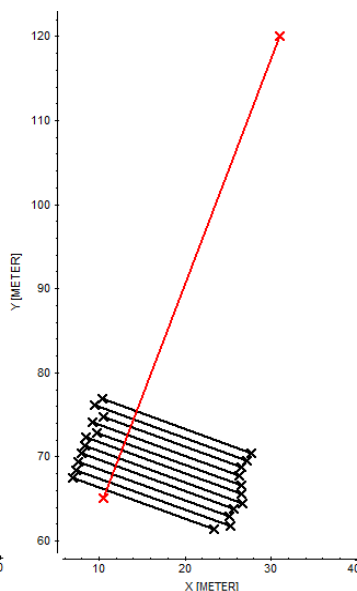
mkn-file: ASCII-file with up to 9 different comment markers defined at the beginning and then each line containing the tracenummer and the code of the marker (2. or 3. colum). The markers are written out as comment marker with the comments defined within the first nine lines.

The gps-data may be loaded using the option RAMAC-GPS. If corc-file is activated a *.corc file is searched consisting of cartesian coordinates.

The option **correct for baseline** allows to correct the coordinates of parallel 2D profiles based on a baseline. An ASCII file with the extension obm must exist which includes the actual cross positions and angles of the profiles at the baseline. Following an example of this ASCII file is given. The programs calculates the x- and y start and end coordinates from the obm file and saves them within the fileheader. An automatic flipping is done if the resulting start coordinate in profile direction is larger than the end coordinate. It is recommended to update the traceheaders based on the fileheader coordinates.

Example:

```
VERSION:1
NUMBER_OF_BASELINES:1
BASELINE:1
START_X:10.450
START_Y 65.200
STOP_X:30.900
STOP_Y:120.100
NUMBER_OF_PROFILES:11
22706
X_LENGTH:4.100
X_ANGLE:90.000
X_DIST2BASE:1.000
22746
X_LENGTH:15.020
X_ANGLE:270.000
X_DIST2BASE:2.000
22825
X_LENGTH:4.200
X_ANGLE:90.000
X_DIST2BASE:3.000
22858
X_LENGTH:14.700
X_ANGLE:270.000
X_DIST2BASE:4.000
```



It is recommended to check whether the repositioning has been successful. This can easily be done by comparing the different 2D-files using the `plotoptions man`. scaling and show marker. The baseline is shown by a marker and should be at the correct position for each datafile.

MALA RD6-Format: The rd 6 format is used with the MALA CX system for a 3D-grid data acquisition. Three different datafiles must be present: `rhd`, `rad` and `rd6`. The `rd6` file must be chosen when starting the conversion - otherwise a warning message appears. The `rad`-file corresponds to the `rad`-file of the Mala `rd3`-format and contains informations about the data acquisition like sampling rate, The `rhd`-file contains informations about the 3D-grid like the x-size and y-size, the resolution in profile direction and the profile spacing. One or two `rd6` files are present. They have the same filename like the `rad` and `rhd` file but in addition ' _1' for the grid lines in y-direction and ' _2' for the grid lines in x-direction. The 2D-lines building up the 3D-grid are sequentially stored corresponding to the `Reflex3D`-file. It is recommended to use the datatype `3D-const.offset` within the import menu. The `profiledirection` and `profileconstant` will be set automatically corresponding to the choose of the `rd6` file (`profiledirection` 'Y' for file `<name>_1.rd6` and `profiledirection` 'X' for file `<name>_2.rd6`).

With the suboption **flip sorting** activated it is assumed that the data have not been stored sequentially scan (channel) after scan (channel) but first trace of all cuts, then second trace of all cuts. If this holds true you must activate the option. In most cases the option must only be activated for one scan direction.

RADAN-Format: With the option `read trace increment` activated the trace increment is read from the header and the ending coordinate is calculated based on the given starting coordinate, the trace increment and the number of traces. With no option activated the entered manual parameters are valid. The markers are read in depending on the settings of the `min./max. marker` values. Newer systems write the user markers within the associated `dzx`-file. If a `WayPt` of the following form is present within the `dzx`-file user marks will be found:

```
<WayPt> <scan>1030</scan> <mark>User</mark> <name>Mark1</name>
</WayPt>
```

With the suboption **read coordinates** activated the local coordinates stored within the `dzx` file will be used for the definition of the fileheader coordinates.

Example:

```
<WayPt>
<scan>0</scan>
<distance>0.0000000</distance>
<localCoords>0.0000000,0.3000000,0.0000000</localCoords>
</WayPt>
<WayPt>
<scan>231</scan>
<distance>2.3100000</distance>
<localCoords>2.3100000,0.3000000,0.0000000</localCoords>
```

The program extracts the start/end coordinates 0/2.31 and 0.3/0.3 from the local coordinates. The profile direction and constant will be automatically updated based on the coordinates.

The original data are saved either as 8 bit, 16 bit or 32 bit integer values.

The newest GSSI systems (from SIR-30, SIR-4000) use the 32 bit format. These data are normally unprocessed and must be stored within `Reflexw` using the 32 bit floating point output format in order to preserve the full data resolution. The

outputformat is automatically set to 32 bit floating point. A scaling factor smaller than 1 (e.g. 0.01) might be useful in order to use the same plotscale factors as for the 16 bit data afterwards. These newer systems use a gps-file with the extension dzg for the gps synchronization. Use the option **GSSI-GPS** for the import of the GPS-coordinates (option update traceheaders or within the traceheader menu).

A **GPS-tmf** file for the synchronization of the GPR-unit and a GPS-device is supported. A tmf-datafile is created when using the older SIR-3000 or SIR-20 together with a GPS-system. The tmf-datafile is a binary file and it contains:

1. the start and end GPS-positions (in degree) and the start and end GPS UTC-time
2. the timing of each marked scan

The option GPS mark/times controls the parameters which are stored within the Reflexw traceheader output.

With activated option GPS mark/times the GPS-times of each marked scan are stored on TimeCollect in secs after midnight. No coordinates will be stored.

With deactivated option GPS mark/times only the start and end GPS positions and GPS UTC-times are stored on the traceheaders of the first and last trace respectively and the timings of each marked scan will be ignored.

The start and end UTC-times within the tmf-file needn't be identical to the start and stop times of the GPR data acquisition (e.g. a wheel has been used or there is a delay between the data acquisition and the start of the tmf-file). In this case the timings of each marked scan must be used (option GPS mark/times activated). On the other hand it also may happen that the mark timings do not exactly match. Then the use of the overall start and end UTC times might be a better solution. Therefore the user must check if the resulting GPS coordinates coming from the procedure described below do match.

It also might happen that no GPS signal is present. Then no GPS-coordinates and no GPS UTC times are available. If the overall GPS end UTC time is 0 (no signal at the end of the profile) the program automatically determines the GPS end UTC time either from the times of the GPR data acquisition unit if available within the tmf-file or from the number of acquired traces and the number of scans parameter. Corresponding messages are created within the import report.

Based on the imported GPS-times it is possible to input GPS-coordinated data from an external ASCII-file using the type GPS-times or GPS-times spatial interpolation within the traceheader menu (see chap. 1.9.6 for a detailed description) or directly during the import using the option Update traceheaders/gps coordinates (see also chap. 1.5.6).

In order to use this option the following currentization must be done within the traceheader menu or within the update traceheader box:

1. gps-times or gps-times spatial interpolation
2. optional UTM-conversion if necessary
3. optional calculate distances

To be considered: during the RADAN-dataimport with deactivated option GPS mark/times the GPS-start and end coordinates given within the tmf-file are also stored within the traceheader of the first and the last trace respectively. These coordinates will be overwritten when using the option update based on gps-times or gps-times spatial interpolation. The coordinates are written out for completeness and may be used when no external ASCII gps-file is present.

SEGY-Format: Some programs (e.g. Eavesdroper) are writing out 'modified' SEGY-data, e.g. the SEGY-fileheaders or the traceheaders are missing. In order

to be able to import these data REFLEXW allows to neglect the SEG-Y-file- and/or traceheaders (options read fileheader and read traceheader).

If the original SEG-Y raw data have been recorded on a Unix machine the option swap bytes must be activated. The dataformat is not always correctly defined especially for 32 bit swapped (Unix) data. The option IBM controls in this case whether the data are IBM-compatible or not. The program also controls whether the data are Unix based or not and sets the option swap correspondingly. This is not possible for the option IBM which must be always entered manually. If the receiver coordinates are not specified within the traceheader but the source coordinates the receiver coordinates are automatically set to the source coordinates (zero offset data are assumed).

The distance (bytes 37-40) also uses the same scaler factor as for the xy-coordinates.

Some SEG-Y-data (e.g. SEG-YDECO) show swapped header values and non swapped data values and vice versa. Therefore the option swap header has been included which allows to only swap the header values or the data values. If several shots has been chosen as datatype the original field record number will also be stored within the Reflexw traceheader under the ensemble nr..

If 3D-const.offset has been chosen as datatype the crossline number from bytes 193-196 within the SEG-Y-file will be stored under the Reflexw ensemble-nr.

SEG-Y-Proceq-Format: With the option Proceq format the SEG-Y-formatted data are assumed coming from a Proceq GPR system. The timedimension will be automatically set to ns and the option ps/ns timeinc will be activated. The SEG-Y-data exported from the Proceq system show small original data values and therefore a quite large scaling factor (e.g. 10000) is recommended.

Two additional csv files may be present which will be interpreted.

The first one contains some metadata which will be partly interpreted if either the option read traceincr. or read coordinates is activated. Dependent on the type of acquisition (line scan or area scan, single frequency or dual frequency) one csv file is present for each segy-file or only one csv file is present for the whole block of scans of one area. In addition the dual frequency antenna produces two segy-files (_LF and _HF) but only one csv-file.

Example line scan:

Linescan Feb 2021-007_20210727_175020_HF.sgy ->

Linescan Feb 2021-007_20210727_175020.csv

Linescan Feb 2021-007_20210727_175020_LF.sgy ->

Linescan Feb 2021-007_20210727_175020.csv

Example area scan:

Areascan002_L001_20210726_135121_HF.sgy->

Areascan002_20210726_135121.csv

Areascan002_L001_20210726_135121_LF.sgy->

Areascan002_20210726_135121.csv

Areascan002_L002_20210726_135121_HF.sgy->

Areascan002_20210726_135121.csv

Areascan002_L002_20210726_135121_LF.sgy->

Areascan002_20210726_135121.csv

.....

From the filename the program automatically determines whether a linescan or an area scan is present.

With the import option read traceincr. activated only the traceincrement will be read from the csv file. The following parameters are read from the csv file if available:

Repetition Rate [scan/cm] -> traceincrement

Antenna Spacing [cm] -> S/R distance

With the option read coordinates activated the start and end coordinates of the profile as well as the traceincrement are read from the csv file. This holds true both for a line scan and an area scan.

The following parameters are read from the csv file if available:

Repetition Rate [scan/cm] -> traceincrement

Antenna Spacing [cm] -> S/R distance

Start x [m]

Start y [m]

For an area scan the program automatically controls whether the scan is orientated in x- or y-direction and whether the scan must be flipped by comparing the start coordinate and the grid size. In addition it is assumed that the first scans are orientated in y-direction and the last ones in x-direction. The program controls the parameter Line and reads the scan number and compares it to the number defined within the original filename (e.g. L001).

For a line scan the following parameters must be present for an automatic control of the orientation:
 Scan Direction Left to Right -> orientation in x-direction, no flipping
 Scan Direction Right to Left -> orientation in x-direction, flipping
 Scan Bottom To Top -> orientation in y-direction, no flipping
 Scan Top To Bottom -> orientation in y-direction, flipping

If a gps system is available during the acquisition the Proceq system generates a second csv file containing these gps coordinates. The synchronization of the gps coordinates and the gpr data is done based on a timestamp which is stored both within the gps csv file and the SEGY gpr data. The csv file has the same name as the first metadata csv file plus a filename extension GNSS. Reflexw supports two NMEA sentences: \$GNGGA and \$GPGGA. The storing of the gps coordinates within the Reflexw traceheader is done using the option update traceheaders -> GPS-times.

SEG2-Format: If the trace increment is defined in the header this value is read and the ending coordinate is calculated based on the given starting coordinate, the trace increment and the number of traces. If the original SEG2 raw data have been recorded on a Unix machine the option swap bytes must be activated. The not default header word COLOUR will be used and stored within the Reflexw traceheader flag ikomp. Together with the plotoption ikomp this allows you to use the secondary color for the display with a value of 2 for COLOUR.

Infrasense-Format: With the option read coordinates activated the starting and ending coordinates in profile direction are read from the infrasense header. The start coordinate is set to the beginning distance counter divided by the scale factor. The end coordinate is set to the last distance counter divided by the scale factor. Zero traces (marker traces) are automatically omitted as well as the traces before the trace with the beginning distance counter. The trace after a zero (marker) trace receives a marker flag.

With the option read coordinates deactivated the starting and ending coordinates must be entered manually. No traces are omitted.

In both cases the current distance (distance counter / scale factor) is stored within the trace header of each trace. Obviously the data acquired with the Infrasense system may not always be equidistant. This means that the distances stored within each traceheader do not exactly correspond to the distances calculated from the fileheader coordinates even if the option read coordinates is activated. In a subsequent processing step a file with equidistant traces can be created based on these traceheader distances (see processing step: make equidist. traces).

IDS-Format: There are four different possibilities to define the geometry. The option read coordinates, read trace increment and anten.array-coord. control it:

- With no of these options activated the entered manual parameters for the start and end coordinates are valid.

- With the option read traceincr. activated the trace increment is read from the header and the ending coordinate is calculated based on the manually given starting coordinate, the trace increment and the number of traces.

- With the option read coordinates activated the starting coordinate in profile direction and the trace increment as well as the coordinate perpendicular to the profile direction (profile constant) are read from the original IDS header.

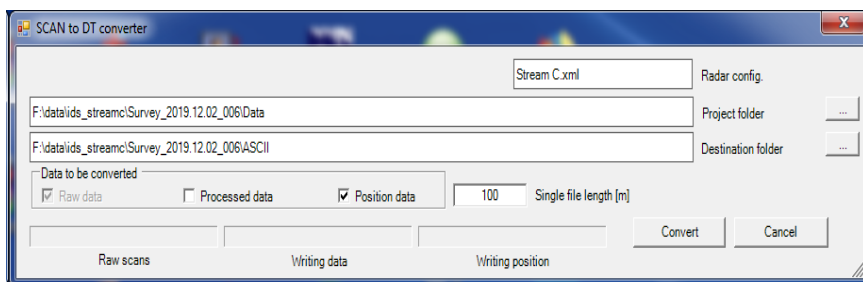
- With the option anten.array-coord. activated the relative antenna positions of an antenna array are taken into account in addition to the overall starting coordinates (see option anten.array-coord. for a detailed description of the parameters).

For most IDS-systems GPS-data are synchronized using a comment marker "gps". Normally the original IDS-file contains these comment markers and they are taken over into the Reflexw file during import. If the IDS-data have been acquired using the Stream software the secondary channels do not contain any informations about the gps-markers. If the conversion sequence has been set to multichannel the gps-markers will be taken over from the first channel data in this case.

Some newer IDS system like the Opera device uses a different format like the dt-format. Reflexw only allows to import dt-files. These files must be generated using the IDS HD-data converter.

Within this data converter the option Position data must be activated if the gps coordinates are also present. The resulting dt-files will be stored within the

destination folder. It is recommended to use the import folder of Reflexw as destination folder.



BITMAP: The amplitude values for the imported REFLEX file are calculated from the color intensity of each bitmap point (sum of the three RGB values divided by 3). Therefore a gray scale bitmap will be best imported. The timeincrement must be entered.

3D-Radar: imports the data from the 3D-Radar Examiner program (default extension vol). The data are 3-dimensional data acquired on parallel 2D-lines. The geometry is normally read from the original data. With the conversion sequence set to "parallel lines" the 3D-Radar file is automatically subdivided into different 2D-Reflexw files. If the filename has been set to "original name" the program automatically adds the number of the 2D-lines to the output filenames - example: the 3D-radar file testa.vol contains 20 different 2D-lines - the Reflexw files will have the names testa000.dat, testa001.dat,..., testa020.dat.

JSF: imports the data from the sonar systems of EdgeTech. Normally the original data are stored as complex data (real and imaginary part). Reflexw only imports the real part of these data.

PS3-Sonne: imports the data from the Paradigma system installed on the ship Sonne. The format is a revised version of SEG Y without the EBCDIC and the 400 byte fileheader. The data are assumed to be 2 byte integer (swapped). The timedelay is assumed to be stored on bytes 193-194 within the 240 byte traceheader and it is multiplied by a factor of 1.3333 in order to convert the depth (meter) values into two-way timedelay ms values. With the option ignore 0-delay traces activated traces with a timedelay of 0 will not be imported.

SEG Y-PRISM: This is a modified SEG Y-format for data from the Prism software from Radar Systems Inc. The main differences to the standard SEG Y-format are: Timeincrement and lag time are given in ps and the lag time is assumed to be negative.

The receiver group elevation (bytes 41-44) are assumed to be 32 bit floating point values and will be stored within the traceheader z-coordinates.

The coordinate units is no standard. If set to 2 a DMS-format is used and not arcsec.

64 bit precision coordinate values are stored within bytes 183-190 and 191-198 (double precision float values) respectively. The option read 64-bit coordinates controls if the 64-bit coordinates will be used instead of the 32 bit values (bytes 73-76 and 77-80). If the coordinate unit (byte 89-90) is "DDMM.SSSS" the coordinates will be automatically converted into degrees.

Within the group X-coordinates (bytes 81-84) the distance coordinates might be stored. With the option store X-rec.group on distance activated the X receiver group coordinate will be stored on the traceheader distances from which also a mean traceincrement and therefore the start and end fileheader coordinates will be calculated.

A marker is stored within bytes 237-238 and 239-240.

GEOSCAN32: The two file formats .gpr and .gpr2 of the OKO-2 GPR from Geotech are supported. With the option read trace increment activated the trace increment is read from the header and the ending coordinate is calculated based on the given starting coordinate, the trace increment and the number of traces. With the option read start coord. the start coordinate will be read from the header in addition.

The import of the 3D-data with the extension prd is also supported. This 3D-files consists of 2D-cuts acquired in y- and x-direction. Reflexw automatically recognizes the individual 2D-scan and the scan-direction and sets the corresponding traceheader parameters (ensemble nr. and scan direction). The traceheader coordinates are also automatically updated. A Reflexw 3D-datafile may be generated within the 3D-datainterpretation using the type "use independent interpolation scheme for pure x- and y-2D-lines" based on traceheader coordinates.

UWB-MEDOAT: imports the data from the Meodat gpr-system. The data are blocked and contain 2 channels. The channels will be automatically demultiplexed and stored on two different datafiles. The filenames will get an extension corresponding to the entered number within the fileheader coordinates panel. Example: the number is set to 0 and original name is used for the filename specification. Then the imported data of the first channel has the filename file1_00.dat and the second channel file1_01.dat.

The number of samples is 511 or 4095. The timeincrement is fixed by 0.1 ns (not given within the original data).

DECOD: imports the data from the Decod gpr-system. The data are blocked and contain several channels. The channels will be automatically demultiplexed and stored on different datafiles. The filenames will get an extension corresponding to the entered number within the fileheader coordinates panel. Example: the number is set to 0 and original name is used for the filename specification. Then the imported data of the first channel has the filename file1_00.dat and the second channel file1_01.dat.

GEOSCANNERS-GSF: import the data from the Geoscanners GPR-system. Reflexw supports multichannel data. Activate the option **flip channels** if the import of the dual channel data is not correct (channel 1 and 2 are flipped and mixed).

IMPULSERADAR: allows to import the data from the ImpulseRadar-systems.

The dataformat may either be 16 bit or 32 bit. The outputformat is automatically set to 32 bit floating point if 32 bit data are present. In this case a scaling factor smaller than 1 (e.g. 0.0001) might be useful in order to use the same plotscale factors as for the 16 bit data afterwards. The gps-data may be loaded using the option RAMAC-GPS. Reflexw supports the import from the multichannel **Raptor** data. In this case a single gps-file for one swath is present together with a geometry file for the different channels (extension ord). Reflexw automatically reads this geometry file and creates a CorrectOffset.txt file which is used in order to correct the gps coordinates for the different channels. For this purpose the conversion sequence should be set to **multichannel** and the options **Utm-conversion** and **correct for offset** must be activated.

The screenshot shows the 'Reflexw dataimport' dialog box. Key settings include:

- Fileheader-coordinates:** DistanceDimen: METER, data type: const.offset, ProfileDirection: X, ProfileConstant: Y, XStart: 0, XEnd: 54.03777958, YStart: 0, YEnd: 0, ZStart: 0, ZEnd: 0, number: 0.
- filename specification:** specification: original name, prefix: (empty), filename factor: 1.
- Time and comment specification:** TimeDimension: ns, sample number: 450, time increment: 0.1953125, timerange: 07.89063, file header: 1451, trace header: 40.
- ConversionMode:** conversion sequence: multichannel, max.traces/file: 1048576, swath offset: 0, check.tracelengt: checked, tracent./2D-line: 0, move receivers: checked, antenna offset: 0, move shots: checked.
- format specification:** input format: IMPULSERADAR, output format: new 16 bit integer, scaling: 1.
- ControlOptions:** control format: checked, read traceincr.: unchecked, fix endcoord.: unchecked, read coordinates: checked, read starttime: unchecked, swap bytes: unchecked, 16 bit data: checked.
- update traceheaders/gps coordinates:** use data folder: unchecked, update traceheaders: RAMAC-GPS, 1.original coord.: x-coord, x-coord: checked, y-coord: unchecked.
- utm-conversion:** UTM-conversion: checked, calculate distances: unchecked, correct for offset: checked, use correctoffset.txt: checked, smooth dir: back, middle, forward (10).
- ControlPanel:** Convert to Reflex: checked, apply processing flow: unchecked, CheckExistingFiles: unchecked, check data for NAN: checked, PrimaryFile: selected, SecondaryFile: unselected, Help, data, Exit buttons.

ASCII-MATRIX-FREQ: The original data are stored within the frequency domain using the following format (csv):

1. row - all frequencies -> number of frequencies
2. row - first frequency values for all positions for the imaginary part

.. (number of frequencies.+1) row - last frequency values for all positions for the imaginary part

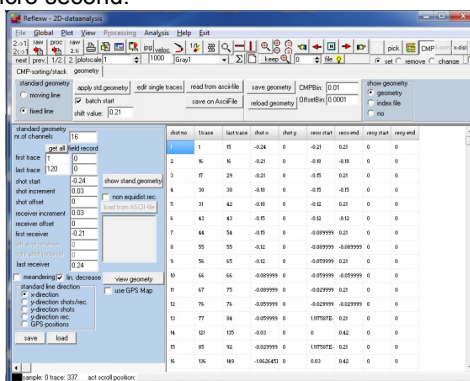
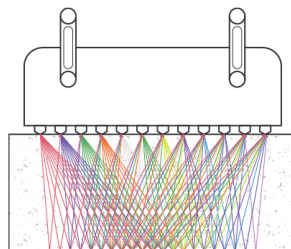
(number of frequencies+2). row - first frequency values for all positions for the real part

.. (2*number of frequencies+1).row - last frequency values for all positions for the y-part

The program automatically perform a Fourier transform in order to transform the data into the time domain.

ASCII-Proceq-PD80: The original data are stored using the ASCII-format of the Proceq PD 80 ultrasound device. Reflexw searches the raw data of the individual channels of the single shots. The proceq device consists of either 8 or 16 channels and allows to acquire data along a single line or perpendicular to the channel arrangement. 15 or 7 shots will be acquired within one single measurement. For the first shot 15 (7) receiver channels right to the shot position will be acquired, for the next shot only the 14 channels right to the shot, for the next only 13 ,..., for the last shot only 1 channel will be stored. No geometry informations will be taken from the original data. Only the sample number is read from the original data. The time dimension is set to micro sec by default. The timeincrement is assumed to be 1 micro second.

The data acquired along a single line may be processed in the same way like a standard 2D seismic reflection survey. The geometry may be entered within the CMP processing module (chap. 1.12.4.1). Here a special option named lin. decrease has been included which takes into account the decreasing number of channels with increasing shot number. A batch start allows to enter the geometry for different scan positions by entering a fixed shift value.



SARA DRM: import format from the drm-files of Sara electronics. The following header words are read in:

sps: samples per second

numero campioni: number of samples

shot x - shot position

coordinata x - receiver position

The fileheader is assumed to have a size of 50000 bytes. The original data are saved as 32 bit integer values. With the option read coordinates activated the fileheader start and end coordinates are defined based on the original coordinates (see above) and in case of a smaller end coordinate than the start coordinate an automatic flipping is done.

HCD-Format: The format supports both 16 bit and 32 bit original data. The datavalues of the 32 bit-data normally exhibit quite large values. Therefore it is recommended to rescale the data during the import using a small scaling values of e.g. 0.00002. This ensures that the resulting datavalues fall into the "normal" 16 bit integer data range. In any case the output format should be set to new 32 bit floating point in order to maintain the full resolution. The program controls this and automatically uses the new 32 bit floating point format.

With the option read trace increment activated the trace increment is read from the header and the ending coordinate is calculated based on the given starting

coordinate, the trace increment and the number of traces. With the option deactivated the entered manual parameters are valid.

The program supports a marker file (same filename but different extension hcm): ASCII file with each line containing the tracenummer of the marker. The markers are written out as distance markers.

General considerations:

Choosing the distance dimension TRACENR. the trace increment is automatically set to 1.

If you want to convert parallel profiles with constant distance between and equal start coordinates in profile direction, the selection of the so called conversion sequence (see also Import ConversionMode) is recommended. This option should also be selected, if you want to convert several shots or CMPs of a certain geometry within a profile line. If the distance between the single profiles is smaller than 1 unit, you have to adjust the filenamefactor accordingly, so that different labels are possible for the automatic naming.

After the setting (please note that, according to the read format additional parameters such as signal length or time increment have to be specified) you can start the conversion of the external data by the option convert to REFLEX. After selecting convert to REFLEX all files existing in the directory ASCII are listed. If you have set the option conversion sequence to no, you have to select a file, which then is automatically converted and stored in the directory ROHDATA. If the target file already exists, a corresponding message is issued. After terminating the conversion the currently converted file is automatically shown either as primary or secondary file.

1.5.1 Import Fileheader-coordinates

Within this group box the fileheader-coordinates must be entered. The geometry of the profile is defined by the 3D-starting coordinates and the 3D-ending coordinates. If the data are acquired in one single plane, the third coordinate (e.g. the z-coordinates) has not to be defined. If the coordinates are already defined in the fileheader of the raw data, the coordinates in profile direction are automatically updated provided that the corresponding ControlOptions are set.

DistanceDimen.: input of the desired distance-dimension. Possible inputs are MM, CM, METER, INCH, FOOT or TRACNR. Choosing TRACNR. means that the trace increment is automatically set to 1 and the name of the axis in profile direction is set to LOCATIONNUMBER.

data type: There are seven different types: const. offset, single shot, several shots, 3D-const.offset, timeslice, single shot/boreholes and single shot/VSP. The data type **,const. offset'** can be used for the import if the measurement is done along a line using a fixed offset between shot and receiver (e.g. Constant-Offset (CO) radar data).

If the data set is the result of a measurement with only one shot point but many receivers in a line, the use of the default data type **,single shot'** data is recommended (e.g. Common-Mid-Point (CMP) radar data, refraction seismic data). **Rec.start** and **rec.end** mean the start and end positions of the receivers along the profile line. **Lat.Offset** means the y-offset (perpendicular to the profile line) of the receivers. **Shot position** means the position of the shot along the profile line. **Shot lat.offset** means the y-offset (perpendicular to the profile line) of the shot. If no conversion sequence is used and after having converted the original shotfile the edit traceheader coordinates tabella menu opens and you may synchronize the overall fileheader coordinates with the traceheader coordinates which will be used in further interpretation steps like first arrival inversion (wavefront inversion or tomography). By default the shotx and recx positions within the traceheaders describe the location along the surface and the shoty and recy positions describe the y-offsets.

In case of data, which contain any kind of multiple shot / multiple receiver data, the default data type **,several shots'** should be used (e.g. reflection seismic - one data set combined of individual CMP data sets).

The data type **,3D-const.offset'** should be used for a 3D-data file consisting of several parallel 2D-lines (see also 3D-datainterpretation). This datatype should be used in combination with the conversion mode 3D-file equidistant or combine lines. The ensemble nr. within the REFLEXW traceheader controls the storing of the individual 2D-lines. All traces belonging to one 2D-line have the same ensemble-number which is stored within the REFLEXW traceheader (see chap. 1.9.6).

The data type **,timeslice'** should be used if the original data are timeslices i.e. time constant cuts within the XY-plane. This data type is also automatically used if the timeslices are created within REFLEXW (see chap. 3.6). The profiledirection should be X and the profileconstant Z. The profileconstant coordinate defines the position of the timeslice in time(z)-direction. Whereby the coordinates in x-direction are given corresponding to the standard const.offset type data, the coordinates in y-direction are defined like the parameters in time-direction for the const.offset data. Therefore the increment in y-direction is given by the y-increment which corresponds to the timeincrement for the const.offset type data and the start position in y-direction corresponds to the starttime for the

const.offset type data and is taken from the original data if the option read starttime is activated. Otherwise it is set to 0 and may be entered afterwards within the fileheader menu.

The data type **single shot/boreholes** can be used for a borehole/borehole geometry. Offset means the offset of the receiver borehole along the surface. Rec.start and rec.end mean the start and end positions of the receivers within the borehole. Shot-offset means the offset of the shot borehole along the surface. Shot position means the position of the shot within the borehole. If no conversion sequence is used and after having converted the original shotfile the edit traceheader coordinates tabella menu opens and you may synchronize the overall fileheader coordinates with the traceheader coordinates which will be used in further interpretation steps like first arrival inversion (tomography). By default the shotx and recx positions within the traceheaders describe the location along the surface and the shoty and recy positions describe the locations within the boreholes. The conversion sequence inline lines may be used in order to update the shot position for different shots (line distance means the increment of the individual shots in this case). For a fixed receiver line and moving shots e.g. activate move shots and deactivate move receivers and enter the shot increment under line distance.

The data type **single shot/VSP** can be used for a borehole/surface geometry. The receivers are assumed to be placed within the borehole and the shot is placed at the surface. Offset means the offset of the receiver borehole along the surface. Rec.start and rec.end mean the start and end positions of the receivers within the borehole. Shot position means the position of the shot along the surface. If no conversion sequence is used and after having converted the original shotfile the edit traceheader coordinates tabella menu opens and you may synchronize the overall fileheader coordinates with the traceheader coordinates which will be used in further interpretation steps like first arrival inversion (tomography). By default the shotx and recx positions within the traceheaders describe the location along the surface and the shoty and recy positions describe the locations within the boreholes. The conversion sequence inline lines may be used in order to update the shot position for different shots (line distance means the increment of the individual shots in this case). For a fixed receiver line and moving shots e.g. activate move shots and deactivate move receivers and enter the shot increment under line distance.

The following options are valid for the data type const.offset:

ProfileDirection: enter the axis in which the profile is orientated. Possible Inputs are X, Y or Z.

ProfileConstant: enter the axis perpendicular to the profile orientation. Possible Inputs are X, Y or Z. With the setting of the ProfileDirection and the ProfileConst the so called profile plane is determined.

XStart: enter the starting coordinate in x-direction.

XEnd: enter the ending coordinate in x-direction.

YStart: enter the starting coordinate in y-direction.

YEnd: enter the ending coordinate in y-direction.

ZStart: enter the starting coordinate in z-direction.

ZEnd: enter the ending coordinate in z-direction.

The following options are valid for the data type single shot:

rec.start: enter the start coordinate of the receiver line.

rec.end: enter the end coordinate of the receiver line.

lat.offset: enter the lateral offset of the receiver line.

shot-pos: enter the shot coordinate.

shot.lat.offset: enter the lateral offset of the shot.

The following options are valid for the data type several shots:

increment: enter the mean traceincrement.

The following options are valid for the data type timeslice:

ProfileDirection: enter the x-axis for the timeslice - choose X.

ProfileConstant: enter the time (z)-axis - choose Z.

XStart: enter the starting coordinate in x-direction.

XEnd: enter the ending coordinate in x-direction.

ZStart: enter the starting coordinate in time(z)-direction.

ZEnd: enter the ending coordinate in time(z)-direction.

The following options are valid for all data types.

number: enter a number between 0 and 99 which is used for the automatic naming of the profile if the filename specification is set to automatic name.

The **X-,Y- and Z-start- and end-coordinates** determine the location of the profile line. However, this plane doesn't have to coincide necessarily with a plane spanned by the coordinate axes. The profiles within a plane don't have to coincide with a coordinate axis either, so that with the setting of the 6 coordinates a profile lying arbitrarily in space is specified. The determination of the so called profile plane, profile-direction and -constant is important for an interactive profile selection and three-dimensional data viewing processes.

In profile direction the end coordinate must not to be identical with the start coordinate. If the end-coordinate is smaller than the start-coordinate the profile is inverted on request after termination of data recording. Start- and end-coordinate together with the number of recorded traces determine the so called trace

increment, which is written in the file header. Though, this increment can be changed again if necessary.

1.5.2 Import filename specification

This group box allows to specify the naming of the converted profiles.

specification: this option allows you to specify the naming of the converted profiles. Entering **automatic name** means that the naming is based on the entered profile coordinates in profile constant direction (see below). Choosing **manual input** means that the filename must be entered manually. Choosing **original name** means that the original name of the raw data is automatically used. A **prefix** may be entered which will be placed in front of the original file name. Choosing **manual /automatic** means that the naming is based both on the manually entered filename and the entered profile coordinates (see below). Choosing **automatic long name** allows a filename specification based on the fileheader coordinates taken into account up to 7 places and 2 decimal places. Thereby also UTM-coordinates can be handled.

To be considered: the max. number of characters for the filename is restricted to 16 (without extension). If the original name exceeds this number the filename is automatically truncated.

filename: enter a name of the converted file provided that the specification is set to manual input.

prefix: enter a prefix which will be placed in front of the original file name.

Filenamefactor: used if the specification is set to automatic name. The start-coordinate in profile-constant direction is multiplied with that factor for naming. A value larger than 1 (e.g. 10) is for example reasonable if floating point numbers should enter the naming. The current coordinate is of course not influenced by this value.

automatic name: The first two digits of the name are taken from the ProfileConstant direction. The start-coordinate in the direction of the profile-constant (only Integer number) is used for the digits 3 to 6. For this the coordinate is being multiplied by the so called filenamefactor. The 7th and 8th digit in the file name is determined by the number.

an example: ProfileDirection is set to Y and ProfileConstant is set to X. The profile extends in Y-direction from 0 to 100 meters. The ProfileConstant coordinates are set to XStart = 1 and XEnd=1. The number is set to 0. With automatic file naming the name will be XP000100.DAT (filenamefactor set to 1).

manual /automatic1: the naming is based both on the manually entered filename and the entered profile coordinates. For the manual name up to 30 characters are permitted. If you use less than 2 characters, the name is filled with '_'. The following 6 digits are taken from the start-coordinate of the profile-constant direction (only Integer number). For this the coordinate is being multiplied by the so called filenamefactor.

manual /automatic2: the naming is based both on the manually entered filename and an automatically determined part. For the manual name up to 38 characters are permitted. If you use less than 2 characters, the name is filled with

'_'. The following 6 digits are taken from the start-coordinate of the profile-constant direction (only Integer number). For this the coordinate is being multiplied by the so called filenamefactor. The 7th and 8th digit of the name are according to the number, as explained under automatic name.

original name: The option **reverse dot search** controls how dots within the original filename are handled. If deactivated the program searches the first dot and only takes the filename part before this dot for the Reflexw filename. If activated the dots which are normally used for the determination of a file extension will be searched from the end of the filename instead from the beginning. This might be useful if a date separated by dots is used for the original filename. The dots within the original name will be automatically replaced by an underline character. Example:

original filename 29.11.2022 144323_001_A01.iprb

With deactivated option the Reflexw filename will be: 29_____.dat

With activated option the Reflexw filename is: 29_11_2022 144323_001_A01.dat

To be considered: Multichannel multiplexed data (e.g. RADAN multichannel files) are automatically demultiplexed. If multichannel data are present the number and therefore the 7 and 8. digit of the filename is used for discriminating between the different demultiplexed datafiles independently from the filename specification. The current number is increased by 1 for the second channel and so on.

an example: the original filename is file1; 4 channels are used; the number is set to 0 and original name is used for the filename specification. Then the imported data of the first channel has the filename file1_00.dat, the second file1_01.dat, the third file1_02.dat and the fourth file1_03.dat.

1.5.3 Import ControlOptions

This group box allows to control the conversion and to specify if the coordinates defined in the raw data header are read in or not.

read starttime: Specification not available for all formats. Active means that the start time of the signal available in the original data is considered and saved within the Reflexw fileheader (option start time). Not active means no consideration (start time is set to zero). Please note that in the case of activated it might happen that you must change manually the start time under the option FILE/FILEHEADER in the main menu after having done a special processing step (e.g. static correction).

To be considered for SEG-Y- and SEG2-format: The traceheader timedelays are loaded from from the 240 byte SEG-Y-traceheaders or from the SEG2-traceheaders. If read starttime has been activated the overall starttime of the fileheader is determined from the min. value of all existing individual time delay values. It is possible to correct for the individual timedelays using the processing option static correction with suboption load trace delays.

read coordinates: specification only relevant for some data formats. Active means, that the start- and final coordinates in profile direction available in the PULSEEKKO-data or the final coordinate in profile direction available in the MALA RD3-data (the start coordinate is automatically set to 0) are automatically considered (the coordinates in profile constant must still be entered manually). The mean trace increment is calculated based on these values and the number of recorded traces. Concerning the PULSEEKKO-data in addition the distance coordinates of each trace are stored in the corresponding traceheaders. For RADAN data the local coordinates stored within the dzx file will be used for the definition of the fileheader coordinates.

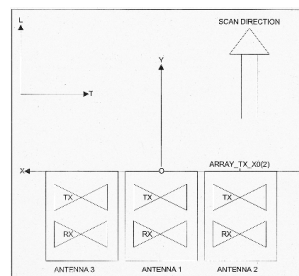
Activating the option TraceHeaderDistances in the PlotOptions menu allows you to plot the data (only wiggle mode) based on these coordinates, which have not to be equidistant.

IDS data: For IDS data the start coordinates in profile direction and perpendicular to it are taken into account (code FX and AM).

Deactive means no consideration of the original coordinates - the coordinates in profile direction, which are manually entered, will be considered.

There are different options which control the type of data acquisition. Depending on the IDS-system the coordinates of the transmitter and receiver antenna are stored in a different way. Therefore an option named **old IDS-system** has been included. It must be controlled manually if the option shall be activated or not. With old IDS-system activated another option named **anten.array-coord.** will be enabled.

The relative coordinates of an antenna array will be read in either if old IDS system is deactivated or if old IDS system is activated together with anten.array-coord.. If old system is activated and anten.array deactivated the relative antenna positions are not considered but only an overall position in profile direction and constant.



The following codes will be considered:

FI: if set to 1 the line will be automatically flipped

FX: scan offset - distance between scan starting and reference point. If FI is set to 1 (flipped profile) this value also points to the overall scan starting after flipping. Therefore for a meandering profile acquisition this value should be equal for all lines.

AM: array direction L or T and start coordinate perpendicular to scan direction (example: AML 5.500000E-01 5.500000E-01 or AMT 5.500000E-01 5.500000E-01)

Two types of antenna arrays are taken into account:

ATR: x-coord., y-coord., angle, frequency for both transmitter and receiver. The x-coord. will be added to the profile constant start coordinate given in code AM and the y-coord. will be added to the profile start coordinate given in code FX.

Example:

ATR 3.330000E-01 2.500000E-01 9.000000E+01 2000 3.670000E-01 2.500000E-01 9.000000E+01 2000

Older systems use another code including the complete geometry of the antenna array in two separate code blocks using the same format as given in code ATR: transmitter array ATX, example:

ATX -1.650000E-01 4.500000E-02 0.000000E+00 1600 -5.500000E-02 -4.500000E-02 0.000000E+00
1600 5.500000E-02 -4.500000E-02 0.000000E+00 1600 1.650000E-01 -4.500000E-02 0.000000E+00
1600

Receiver array ARX, example:

ARX -1.650000E-01 -9.500000E-02 0.000000E+00 1600 -5.500000E-02 -9.500000E-02 0.000000E+00
1600 5.500000E-02 -9.500000E-02 0.000000E+00 1600 1.650000E-01 -9.500000E-02 0.000000E+00
1600

read trace increment: input only for some data formats. Active means, that the trace increment is read from the fileheader if not equal 0. Based on this trace increment and on the given start coordinate the final coordinate of the line is redefined. Deactive means no consideration - the trace increment is calculated from the start- and final coordinates and from the given number of traces.

SEGY-data: this format does not directly specify the traceincrement. Reflexw offers two possibilities:

a. the traceheader word "TRACE INTERVAL" has been defined within the EBCDIC header - then the value behind this traceheader word will be taken over.

Example: C34 TRACE INTERVAL 25

b. the traceincrement will be read from the file with the extension csv (same filename as the original SEGY file) which contains a header word Repetition Rate which will be transformed into the traceincrement. Such a file will be generated from the export tool from **Proceq**. In addition the source receiver distance will also be taken from this file (header word Antenna spacing).

c. the mean traceincrement will be determined from the distance between the first and last trace which has been calculated from the distances defined within the SEGY-traceheaders (bytes 037-040) if available.

ignore 2.traceincr.: input only for RADAN-data - if activated the traceincrements stored within the secondary headers of a multichannel dataset will be ignored.

The multichannel GSSI systems allow the input of a different traceincrement for a multichannel array. It might occur that the traceincrements stored for the secondary channels (channel 2, ...) are not correct (the normal situation is that the increments are all the same). If the option is activated the traceincrement of the first channel will be used for all channels.

fix endcoord.: if activated and activated option read trace increment the entered endcoordinate in profiledirection is fixed and the startcoordinate is changed based on the given endcoordinate and the given traceincrement.

NOTE: If the entered startcoordinate is bigger than the endcoordinate an automatic flipping is done in any case (also if the option meandering is activated for the conversion sequence parallel lines) and the startcoordinate becomes the endcoordinate and vice versa.

Therefore in the case of bigger startcoordinate the entered startcoordinate is fixed if the option fix endcoord. is activated and with deactivated option the entered endcoordinate will be fixed.

read stacknumber: specification only for BGREMR-data. YES means, that the data are normalized based on the stacknumber stored in the BGREMR-fileheader: 16*Data[i]/Stacknumber. NO means no consideration of the stacknumber - direct transfer of the data.

swap bytes: specification only for SEG2- and SEGY-data. Activate this option if the bytes shall be swapped for the conversion. This hold true if the original raw data have been recording on a Unix machine.

IBM: specification only for SEGY-data. The option controls if the 32 bit SEGY-data are IBM compatible. The option is only active with swap bytes activated. Normally you should deactivate the option when the data are coming from an PC-Unix system and activate this option for e.g. older Unix systems but it is not possible to give general rules.

unsigned: specification only for SEGY-data. If activated 16 bit integer SEGY data (format code = 3) are assumed to be unsigned. Data from the Coda system may have been stored as unsigned 16 bit data. It must be controlled manually if this holds true as no unique informations are given within the Coda SEGY-header.

Swap header: specification only for SEGY-data. By default the option is set to actual settings of swap bytes. The option has been included because some SEGY-data show swapped header values and non swapped data values and vice versa. Activating this option (SEGy-header values will be swapped) and deactivating the option swap bytes (SEGy-data values will not be swapped) might be useful for e.g. SEGyDECO data.

Use sweeplength: specification only for SEGY-data. If activated the sweeplength stored within the SEGy-fileheader is used instead of the sample number in order to determine the length of the output trace

control format: activate this option if you which the program to control the input format during conversion. In that case the program performs a so called plausibility test, i.e. various values (e.g. the number of samples per trace) are checked for reasonable values. If an "incorrect" value is recognized (e.g. number of samples smaller than 0 or larger than 16000), the conversion is terminated with the corresponding message.

read fileheader: only for SEGy-data. By default activated. Deactivate this option if the original data do not contain the SEGy-fileheaders (3200 EBCDIC-header and 400 byte binary header), e.g. data generated by the program Eavesdroper. If deactivated you must specify the data formattype manually from the following format codes:

4 byte ibm

4 byte integer
 2 byte integer
 4 byte IEEE
 1 byte integer

read traceheader: only for SEGY-data. By default activated. Deactivate this option if the original data do not contain the SEGY-traceheaders.

ignore blocksize: only for SEG2-data. By default activated. If activated the program ignores the datablocksize within the SEG2-traceheader and only uses two special bytes for discriminating the individual traces. If deactivated the datablocksize is also taken into account. The option has been included because sometimes the datablocksize is not correct and sometimes the two special check bytes are missing.

Corr.ampl.: only for SEG2- and Yokogawa-data. If activated the descaling and stack factors stored within the original data are used to correct the data values.

ignore stringlengths: if activated the trace strings will be interpreted even if the stringlengths are set to 0 (a different method for discriminating the individual SEG2-traceheader strings will be used). Activate the option if problems occur during reading the SEG2-traceheaders ,e.g. no timeincrement has been detected.

ps/ns timeincr.: only for SEGY-data. Two cases must be considered:
 Timedimension set to ns: If activated the segy-timeincrement is assumed to be in picosec. The option should be activated for ns-data because SEGY only allows to store the timeincrement as an integer value. If the option is deactivated the program stores the timeincrement in μ s which will be 0 in most ns-timeincrement cases because of this integer restriction.
 Timedimension set to ms or μ s: If activated the segy-timeincrement is assumed to be in nsec. This might be the case for e.g. high frequency ultrasound data.

SEGY-coordinates groupbox:

coord. in degree: only for SEGY-, JSF- and PS3 data. If activated the traceheadercoordinates given in arc secs within the SEGY- or JSF-file or PS3-file are transformed into degree values.

Ignore scale: if activated the scaler for the coordinates and the elevations within the SEGY-traceheader will be ignored. You can enter a new scaling factor for both the coordinates and the elevations (options **coord.scaler** and **elevat.scalier**). By default these manual scaling values are set to 1.

unsigned: specification only for SEGY-data. If activated the SEGY traceheader coordinates (bytes 73-88) are assumed to be 32 unsigned. Data from the Knudsen Engineering system may have been stored as unsigned 32 bit data (no SEGY standard).

Ignore group: specification only for SEGY-data. if activated the source coordinates are also used for the receivers - the original group coordinates will be ignored.

calculate traceincr.: only for SEGY-data. If activated the mean traceincrement will be calculated from the summed up distances derived from the original SEGY-receiver coordinates. The end coordinate in profile direction will be updated

accordingly. The actual profile distance will be stored within the Reflexw traceheader.

man. samplenr.: only for SEGY-data. If activated the sample number within the SEGY-fileheader is ignored and the manually entered value is used instead.

use sweeplength: only for SEGY-data. If activated the sweeplength stored within the SEGY-fileheader is used instead of the sample number in order to determine the length of the output trace.

read marker: only for SEGY-data. If activated marker events stored within the SEGY-traceheaders are used as comment markers. The event numbers are stored on bytes 193-196 within the SEGY-traceheader and a marker is set at that position where the event number changes (at the 1. trace a marker is recognized if the event number is ≤ 0). The number is taken as the comment.

GPS mark/times: only for RADAN-data. If deactivated the marker timings within the tmf-file are not taken into account.

min.marker: only for importing 16 bit or 8 bit RADAN-data. Enter a 16 bit value without sign (between 0 and 65535) or a 8 bit value respectively depending on the dataformat of the original RADAN data. The program also controls if 16 or 8 bit data are present and changes the min./max. markers values if they do not fit the original Radan fileformat. The parameter controls the minimum value of the second byte within each trace of the RADAN-data for detecting a marker. Change this value by trial and error if some markers could not be found. The following default values can be given:

57471: SIR-10, SIR-2, SIR-2000 16 bit data

20000: SIR-20, SIR-3000 16 bit data

220: SIR-10, SIR-2, SIR-2000 8 bit data

90: SIR-20, SIR-3000 8 bit data

The very new GSSI systems (e.g. SIR 3000) automatically set markers every 1 m. In order to not consider these markers you may change the min.marker value in a special way. Example: 25700 for 16 bit data or 110 for 8 bit data but this may not be always correct.

max.marker: only for importing 16 bit or 8 bit RADAN-data. Enter a 16 bit value without sign (between 0 and 65535) or a 8 bit value respectively depending on the dataformat of the original RADAN data. The program also controls if 16 or 8 bit data are present and changes the min./max. markers values if they do not fit the original Radan fileformat. The parameter controls the maximum value of the second byte within each trace of the RADAN-data for detecting a marker. Change this value by trial and error if some markers could not be found.

61184: SIR-10, SIR-2, SIR-2000 16 bit data

30000: SIR-20 16 bit data

239: SIR-10, SIR-2, SIR-2000 8 bit data

127: SIR-20, SIR-3000 8 bit data

marker autodetect: if activated the min./max. marker values will be automatically determined from the RADAN-data format.

no 32bit-systemmarker: if activated the systemmarkers for 32Bit Radan data will be ignored. The system marker within the original Radan data must have the value -469762048.

UseDZXMarkers: if activated the markers are read from the dzx-file if existing, if deactivated or not existing from the dzf-file

ignore 0-delay traces: if activated traces with a timedelay of 0 will not be imported for the PS3-Sonne format.

correct for baseline: only for Mala rd3- and rd7-data. If activated the option allows to correct the start positions of parallel 2D profiles based on a baseline. The **baseline pos.** must be entered. An ASCII file with the extension obm must exist which includes the actual cross positions and angles of the profiles at the baseline. Following an example of this ASCII file is given. The programs performs a shift of the start coordinates of each 2D-profile based on the given cross position at the baseline (parameter X_LENGTH). If meandering has been activated each second 2D-profile will be flipped. In this case the angles within the ASCII-file will be used in order to check whether they are consistent with the datafile choice.

```
VERSION:1
NUMBER_OF_BASELINES:1
BASELINE:1
START_X:      0.000
START_Y:      0.000
STOP_X:       0.000
STOP_Y:       -28.000
NUMBER_OF_PROFILES: 53
OMP20001_A1
X_LENGTH: 13.6461
X_ANGLE:270
X_DIST2BASE: 2.0000
OMP20002_A1
X_LENGTH: 17.5558
X_ANGLE: 90
X_DIST2BASE: 2.5000
OMP20003_A1
X_LENGTH: 14.0220
X_ANGLE:270
X_DIST2BASE: 3.0000
```

It is recommended to check whether the repositioning has been successful. This can easily be done by comparing the different 2D-files using the plotoptions man. scaling and show marker. The baseline is shown by a marker and should be at the correct entered position for each datafile.

read times from file: only for IDS-data - allows to read the gps synchronized times from an external file named inxxxx.txt with xxxx corresponding to the filename which is identical to the last 4 digits of the datafile.

16 bit data: only for IDS-data - if activated the original IDS-data are assumed to be 16 bit data. if deactivated the data are assumed to be 32 bit data.

1.5.4 Import format specification

This option allows to specify the input format and the output format.

input format: Specification of the format of the data to be converted. At the moment the formats SEG1, SEG2, SEG2-RADAR, RADAN, PULSEKKO, MALA RD3, MALA RD6, MALA RD7, IDS, UTSI, INFRASENSE, RADSYS, GEOSCAN32, BGREMR, BGREMRHELI, DMTKR, BGRDHU, BGRBHRU, TRS2000, ABEM, BISON-2, CSP, NWF, NICOLET, YOKOGAWA, FREE 8BIT, FREE 16BIT, FREE 32BIT, FREE 32BIT FP, FREE 64BIT FP BIT, HP-network, UWB-MEODAT, ASCII-1COLUM, ASCII-1COLUM-ALPS, ASCII-3COLUMNS, ASCII-GORILLA, ASCII-MATRIX, ASCII-3D-KOEHLER, BITMAP, FD-total wavefield, 3D-Radar, OSU DZ2, KGS, JSF, PS3-Sonne, Geoscanners_Gsf, VIY and HCD are supported. Please pay attention that the correct format is specified for the conversion, since the program otherwise can be caught in an infinite loop (restarting necessary) or the program is aborted.

For the conversion from the RADAN-8Bit-format the data are multiplied with the constant factor 16. For SEG1-data, recorded under DOS, the swap bytes controloption must be deactivated, if the data have been recorded according to the DOS-convention. In case you don't know the recording convention you have to test both reading in with activated and deactivated option.

The current input-format is automatically saved in the INI-file.

output format: controls the data format of the converted data. Data may be stored in 16 bit integer or 32 bit floating point format. In case the data are stored in 16 bit integer format the scaling value must be taken into account (see below). To be considered: The DOS-version of REFLEX does not support the 32 bit floating point format. Data stored with this format cannot be accessed by the DOS-version.

There are two new output dataformats: new 16 bit integer and new 32 bit floating point. Data stored with this format can only be accessed by REFLEXW version 3 and higher. These new data formats are using double precision (64 bit) for the traceheader coordinates (see also trace header Edit, chap. 1.9.6). This is necessary when storing GPS-coordinates because the resolution of single precision data used with the two other formats is too small for most GPS-coordinates. In addition a gain factor has been included into the traceheader of the new formats which allows to individually adapt the single traces.

scaling: Specification of a value (real input) the original data are multiplied with before the conversion (unless 16 bit integer original SEG1-data). The modified data are converted. This change of the data is necessary when the dynamic range of the REFLEX-internal format (16 bit integer format) is smaller than the one of the original data. The 16 bit integer format only has a dynamic range of 16 Bit, in order to save memory. Data in SEG1- or other formats can be saved as 32 Bit real data or as even larger binary coding. Values larger than 16 Bit therefore are clipped during the conversion, i.e. they are assigned the maximum value of +- 32767.

Some format specifications:

ASCII-1COLUM: ASCII format whereby each line contains one sample (floating point representation). The number of samples per trace must be defined manually as well as the timeincrement and the time dimension.

ASCII-3COLUMNS: ASCII format whereby each line contains the distance, time and amplitude. The overall first line is a header line. The number of samples per trace, the traceincrement and the timeincrement are automatically determined from the data. It is assumed that the data are equidistant in time and the number of samples is equal for all traces.

ASCII-MATRIX: ASCII format whereby each line contains all samples of one trace. Each value (floating point representation) must be separated from another by a sign not containing in the floating point representation, e.g. a blank character. Example (10 samples per trace - 2 traces):

```
200.1 205.1 208.9 180. 170 60. -20. -50. -80. -120.
210.1 215.1 218.9 190. 180 70. -30. 60. -9. -130.
```

The number of samples per trace is automatically read in from the number of values within the first line. The time increment must be entered manually (see Import HeaderInput).

The options file header and trace header allow to enter a number of comment lines at the beginning of the file (option file header) and the number of comment lines at the beginning of each trace line (option trace header). All comment lines are only read in and not interpreted.

ASCII-3D-Koehler: import ASCII format for a 3D-dataacquisition. The datatype within the import menu will be automatically set to 3D-const.offset. The acquisition direction must be in x-direction.

There is an header in front of the data of 25 lines:

```
1..9. line will be overwritten
10.line: xmin [mm]:          145.000
11.line: xmax [mm]:          320.000
12.line: traceincrement:      7.000
13.line: number of x-lines:    26.000
14.line: ymin [mm]:           325.000
15.line: ymax [mm]:           500.000
16.line: profile increment:    7.000
17.line: number of y-lines:    26.000
18..22. line will be overwritten
23.line: timeincrement:        0.031250
24.line: number of samples:    1761
25.line: time measured:        55.031250
26.line .... n.line data
```

Each data line contains a complete trace with data written in ASCII-format.

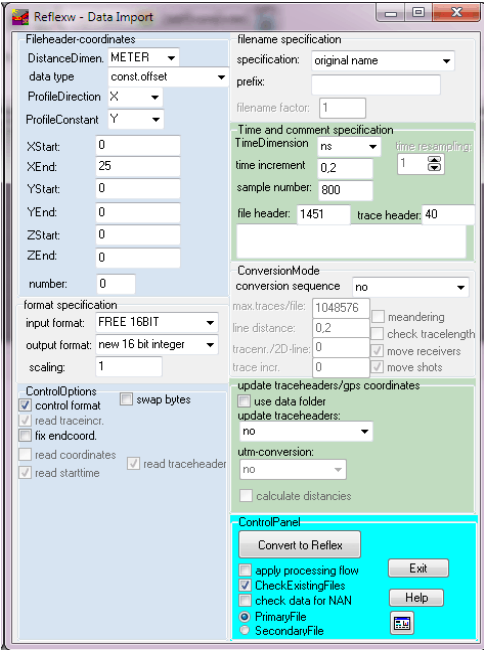
FREE XXBIT: allows to import binary data which have been stored as:

```
8 bit integer (FREE 8BIT)
16 bit integer (FREE 16BIT)
32 bit integer (FREE 32BIT)
32 bit floating point (FREE 32BIT FP)
64 bit floating point (FREE 64BIT FP)
```

The number of samples per trace must be defined manually as well as the timeincrement and the time dimension (see Import HeaderInput).

The options file header and trace header allow to enter a number of bytes at the beginning of the file (option file header) and at the beginning of each trace line (option trace header). All comment bytes are only read in and not interpreted.

The example at the right is for a gsf-file from Gescanners.



1.5.5 Import ConversionMode

this group box allows to specify sequence conversion modes.

The options in the right block serve to specify the input format and the input mode as well as to set the plot parameters:

conversion sequence: This option allows you to import several 2D-lines within one step, e.g. a 3D-data volume consisting of individual parallel 2D-lines. After selecting this option you have the choice between no, combine lines/shots, parallel lines, inline lines, geometry file and 3D-file equidistant. When selecting these options the original profiles remain unchanged and are stored under different names, determined automatically by the program corresponding to the settings of the filename specification.

To be considered: If a conversion sequence is specified there are 2 different possibilities of choosing the datafiles:

1. choose the datafiles from the openfile dialogue: all chosen original data files are sorted with ascending order. Problems may occur if the filenames don't have the same length (e.g. file1, file2,..., file11). In this case the chosen files are resorted in the manner: file1, file11, file2, In this case a warning message appears. Please take care that all original filenames have the same length - for example: file01, file02, ..., file11.

2. open an external ASCII-filelist with the extension ".lst": the external filelist contains all wanted datafiles in an arbitrary order (one row contains one filename). The datafiles must be stored under the same path like the ASCII-filelist. Example:

TEST__02.sg2

TEST__01.sg2

TEST__03.sg2

TEST__04.sg2

Some recommendations for different data:

GPR zero offset data:

data type: const.offset

conversion sequence: no or parallel lines

GPR CMP data:

data type: single shot

conversion sequence: no

Seismic reflection shot data:

data type: several shots

conversion sequence: combine lines/shots

Seismic refraction shot data:

data type: single shot

conversion sequence: no or several lines

borehole-borehole data:

data type: single shot or single shot/boreholes

conversion sequence: no or several lines

borehole-borehole data:

data type: several shots

conversion sequence: combine lines/shots

VSP data:

data type: single shot/VSP

conversion sequence: no or several lines

Combine lines/shots means that all selected profiles are combined in one profile and therefore also in one file in the sequence of selection. Use this selection for a shot gather (see also CMP-processing) or for a 3D-data acquisition (see also 3D-datainterpretation) for example.

With **parallel lines** it is assumed that the selected profiles are parallel profiles with identical start and end coordinates in profile-direction. The prespecified profile-constant is altered automatically by the program by a value to be predefined (option line distance - see below). The automatic naming is according to the determined profile-constant.

With **inline lines** it is assumed that the desired profiles lie on one line each with a constant distance to the other (e.g. single shots or CMPs). The two options move receivers and move shots allow to specify whether the shots and/or the receivers are moved by the constant distance (parameter line distance). For automatic naming each of the two-digit numberings is increased by one starting from the preset numbering. Therefore, there can be up to 100 different profiles converted in one step.

Activating **several lines** no assumption about the geometry will be made for the desired profiles. The option allows to select a number of different original files which will be converted and saved as separated files. The filename specification original name is recommended. This option might be used e.g. in combination with the datatype single shot, single shot/boreholes or single shot/VSP in order to import several single shots (e.g. from a seismic refraction survey) within one step. In this case the program checks whether the trace coordinates are present within the original data and synchronizes automatically the Reflexw traceheader coordinates and the overall fileheader coordinates. If the original data do not contain the coordinates the traceincrement is set to 1 and all lines start at 0.

With **3D-file equidistant** it is assumed that the original profile is a single 3D-file consisting of parallel 2D-lines with equidistant trace increment and equal trace number for each 2D-line and equidistant distance between the individual 2D-lines. The parameters line distance, trace incr. and tracencr./2D-line must be entered (see below).

With **geometry file** the main geometrical data of the files to be converted are read from an additional ASCII-file (name GEOMETRY.*). One line of the ASCII-file must contain the geometry informations as following:

```
name(incl.extension)
profile direction (e.g. X)
profile constant (e.g. Y)
start coordinate in profile direction
end coordinate in profile direction
start coordinate in profile constant
end coordinate in profile constant
output filename (without extension) - only necessary if filename
specification is set to manual input
```

All inputs must be separated by at least one blank or tabulator sign (use tabulator sign if you want to create the geometry file by Excel or Quattro Pro, e.g.).

Example of a geometry file (name: GEOMETRY.1):

```
TEST_01.Dzt X y 0.0 40.0 0 0 test1
TEST_02.Dzt X y 0.0 40.0 1 1 test2
TEST_03.Dzt x y 0.0 40.0 3 3 test3
TEST_04.Dzt y x 10.0 50.0 1.0 2.0 test4
```

If the file is not correct (e.g. an input is missing) or a file name of the files to be converted is missing, a corresponding message is given. The ASCII-file may have more entries than the desired number of files to be converted. The name of the ASCII-file must be GEOMETRY.* and it must be stored under the path ASCII under the current project path.

After clicking on ConvertToReflex first you must select the wanted geometry file and then you are asked to select the wanted original profiles to be imported (multi choice using the ctrl or shift key). After having chosen the wanted profiles the conversion will be done.

With **geometry filelist** the main geometrical data of the files to be converted are read from an additional ASCII-file (name GEOMETRY.*). The format of the ASCII-file corresponds to that one of the option geometry file described above. The name of the ASCII-file must be GEOMETRY.* and it must be stored under the path ASCII under the current project path. In contrast to the option geometry file all files listed within the ASCII-file will be imported even if these are listed twice or still even more. The files must have been stored under the path ASCII under the actual project directory. After clicking on ConvertToReflex first you must select the wanted geometry file and the conversion starts (no files must be selected). If a file within the geometry filelist does not exist the conversion stops with an error message.

With **multichannel** a multichannel importfile (e.g. a RADAN file from the SIR-10) is decomposed into different files and the prespecified profile-constant is altered automatically by the program by a value to be predefined (option antenna inc - see below). The profile constant of the first channel corresponds to the entered profile constant. The automatic naming is according to the determined profile-constant (see also option parallel lines). It is possible to import several multichannel files within one step. The profiles are assumed to be parallel. The option line distance determines the distance between the individual parallel lines. Meandering is also supported. Example of 10 parallel lines with a 2 channel system.

Input: Profileconstant: 100m; antenna inc.: 5 m; line distance: 10 m; meandering active.

This results in the following profile constant values:

- 1. line, 1. channel: 100 m
- 1. line, 2. channel: 105 m
- 2. line, 1. channel: 115 m
- 2. line, 2. channel: 110 m
- 3. line, 1. channel: 120 m
- 3. line, 2. channel: 125 m
- 4. line, 1. channel: 135 m
- 4. line, 2. channel: 130 m

.....

It is possible to enter negative values both for the line distance and/or the antenna inc. and the line distance may not necessarily double the antenna inc. To be considered: a multichannel importfile is always decomposed into different files.

With **multi components** activated a 3-component datafile will be constructed from several original datafiles containing multicomponent data - for further details see 3-component analysis (chap. 1.12.8).

After selecting convert to REFLEX and the possible specification of the profile distance, the profiles to be converted have to be selected by pushing the left mouse button or deselected again if the specification was incorrect. If the start coordinate in profile direction is larger than the end coordinate the line is automatically flipped in profile direction (see also XFlipProfile) after the conversion. To be considered: an update of the traceheader coordinates based on an ASCII-file shouldn't be carried out any more because the original tracenumbers mostly form the base for the traceheader updating.

line distance: Specification of the distance of 2 profiles. This value is used for the automatic coordinate determination for a profile sequence conversion (see above). With combine lines/shots activated the line distance is stored within the fileheader and is used if the combination of the lines results in a 3D-file. With 3D-file equidistant activated the line distance is stored within the fileheader.

antenna inc.: Specification of the distance between the antenna for a multichannel system. This value is used for the automatic coordinate determination for a profile sequence conversion multichannel.

meandering: activate this option if the profiles are acquired meandering. The option is available for the import sequences "parallel Lines" and "combine lines/shots". The meandering performs a flipping of each second original line/shot starting at line/shot number 2 if the entered startcoordinate is smaller than the endcoordinate and starting at line/shot number 1 if the entered startcoordinate is greater than the endcoordinate. With the import sequence "combine lines/shots" activated this corresponds to the processing option XFlip-3Dfile with the flipmode 1 or flipmode 2. To be considered: an update of the traceheader coordinates based on an ASCII-file shouldn't be carried out any more because the original tracenumbers mostly form the base for the traceheader updating.

check tracelength: available only for import format SEG2 and conversion sequence combine lines - if activated the program controls the samplenumber for each original file and sets the output samplenumber to the max. number of samples found.

trace incr.: - parameter only enabled for the conversion sequence 3D-file equidistant - specification of the trace increment within one 2D-line provided that the trace increment is not automatically read from the fileheader of the original file (see also option read trace increment under Import ControlOptions).

tracenr./2D-line: - parameter only enabled for the conversion sequence 3D-file equidistant - specification of the number of traces for each 2D-line within the original 3D-file.

max.traces/file: specification only for Radan, Pulseekko, MALA RD3, IDS and SEG2-data. The max. number of traces in REFLEX is restricted to 1048576. In order to import data files for the input-formats mentioned above with more than 1048576 traces the original profile is automatically subdivided into different blocks with an equal number of traces. These blocks are stored on different files. The parameter max.traces/file controls the size of the individual blocks. The max. value is 1048576 (default value) and is identical to the max. number of traces per REFLEX-file. It is recommended to use a number smaller than 1048576 because

in this case the individual files are easier to handle. If the number of traces to be imported exceed the max.traces/file a new file is automatically created. The starting coordinate in profile direction will be updated and the number is increased by 10. This allows a max. subdivision of the original file into 10 different files with a max. size of 1048576 traces for each file. This results in a maximum size of 1048576 traces of the original file (the number must be set smaller than 10). If a multi-channel data file is imported, the individual channels are also stored on different files with an automatic increase of the geometry number by 1 for each channel (see also peculiarities of various formats - RADAN).

example: original file with 150000 traces and 4 channels (trace increment: 1 cm)

input - max.traces/file: 64000, number: 0

1. line: from 0-640 m with 64000 traces
 - 1.channel number 00
 - 2.channel number 01
 - 3.channel number 02
 - 4.channel number 03
2. line: from 640-1280m with 64000 traces
 - 1.channel number 10
 - 2.channel number 11
 - 3.channel number 12
 - 4.channel number 13
3. line: from 1280-1500 m with 22000 traces
 - 1.channel number 20
 - 2.channel number 21
 - 3.channel number 22
 - 4.channel number 23

If the starting coordinates are greater than the end coordinates the single lines are automatically flipped in x-direction except the case you are using a profile sequence for the conversion. If you have entered profilesequence/inline lines no subdivision into single lines is performed. If the total max. number of 1048576 is reached, the conversion is broken off in this case.

To be considered: the original trace number will be lost. Therefore it is not possible for the secondary segments for example to perform at a later date a GPS-update based on a gps file containing the trace numbers for the synchronization. This GPS-update must be performed directly during the import (see option Update traceheaders/gps coordinates).

1.5.6 Import ControlPanel

This group box controls the conversion.

Convert to REFLEX: Here the conversion of a profile or the profile sequence is started with the preset parameters. After selecting this option and confirmation of the query if the data should be saved under the desired name, all files within the subdirectory ASCII are shown in a window. The desired profile, or for profile sequence-conversion the desired profiles, has (have) to be selected. The current status is shown in a window. The maximum number of traces (Scans) for each recorded profile is limited to 64000 (see ConversionMode). If the number of traces exceeds this value, the conversion is terminated automatically. If the start coordinate of the profile is larger than the end coordinate (in profile direction, i.e. perpendicular to profile-constant), the profile is inverted automatically, if the corresponding query is answered with yes. Such an inversion is necessary if the same profile direction is desired. Each profile can also be inverted in Data processing.

If the data type is set to **single shot** the traceheader coordinates are automatically shown within a **table** after having done the conversion. The coordinates can be changed manually or changed based on the fileheader coordinates (option update from fileheader). On the other hand the option update fileheader allows you to set the fileheader coordinates (rec.start and end, shot position) based on the traceheadercoordinates. The option save changes saves the current settings of the traceheader coordinates. The option reload from file reloads the traceheader coordinates from file.

After the conversion a report is listed including the input and output filename as well as informations about the fileheader coordinates and the sample and trace numbers. This report can be saved on an arbitrary texte file (option save as).

CheckExistingFiles: activate this option if you want to check for existing files before conversion. In that case a warning message appears if the file to be created for conversion already exists.

PrimaryFile: activate this option if you want the converted profile to be the primary file. Option not available for REFLEX DataView.

SecondaryFile: activate this option if you want the converted profile to be the secondary file. Option not available for REFLEX DataView.



enter PlotOptions. Enter this option, if you want to change the plot parameters and to save them in the fileheader of the line to be imported.

Exit button: terminates the import and exits to the main menu.

Help button: activates the main help menu for the data import (Import Menu).

Update traceheaders/gps coordinates:

The option **update traceheaders** allows to automatically update the geophone-, shot- and CMP-coordinates either based on the distance coordinates of the file header (coordinates in profile direction and in profile constant) or based on a GPS ASCII-file. The option is the same like given within the edit several fileheaders menu (see chap. 1.10). The **original trace numbers** are used for the synchronization of the GPS coordinates if a format was used in which the synchronization is based on the trace numbers. This allows processing steps that change the sorting or the number of original trace numbers (e.g. x-flipping in profile direction if the option meandering has been activated) to be carried out before the GPS coordinates are imported. The option **1. original coord.** allows you to specify where the first original coordinates shall be stored - either on the x-traceheader coordinates or on the y-traceheader coordinates.

With the option **use data folder** activated the GPS-files will be searched under the original data folder. Otherwise the GPS-files will be searched under the folder ASCII under the actual project directory.

Pecularity for IDS-GPS data: The 4th character indicating the channel number within the automatically defined GPS-filename will be set to 1 if the corresponding filename does not exist. This might be necessary when multiple channels of data were collected.

The option **utm-conversion** includes the following utm-conversion possibilities:

UTM-conversion

UTM-DEGREE-conversion

RD-conversion (netherlands)

Swiss CH1903

Swiss CH1903+LV95

Swiss CH1903+LV03

These conversions Swiss CH1903+LV95 and Swiss CH1903+LV03 use the local reference system CH1903+ with its reference frame LV95. With the option Swiss CH1903+LV03 an additional transformation of the plane coordinates LV95 to LV03 will be done. In order to use these options the free

ReframeComDll_x86.msi from SwissTopo must have been installed.

The option **latitude** allows to specify where the latitude is stored within the traceheader (latitude in degrees either in x- (shot x, rec x and CMP x-pos) or y-coord). The longitude is automatically vice versa in y- or x-coord.

The option **NMEA-string no degree** allows if activated that the \$GP Strings do not contain degree values but the string will be interpreted as it is.

The option **calculate distancies** allows to update the traceheader distances based on the x- and y-receiver traceheader coordinates. The start distance is always set to 0. The options use z-coordinates and straight line from start point defined within the edit traceheader menu are valid (see chap. 1.9.6).

Activating **smooth xyz-coord.** allows to smooth the traceheader receiver coordinates over a selectable number of traces given by the parameter smooth. For common offset lines the same smoothing will be done for the shot coordinates.

The option **correct for offset** corresponds to the option "correct for offset in profile direction" within the traceheader menu or the edit several fileheaders menu with activated option read from file and allows to correct the xy-GPS coordinates for an offset in profile direction and perpendicular to it. The two correct offsets (in profile direction - offset - and perpendicular to it - lat.offset) are read from an ASCII-file named CorrectOffset.txt which must exist under the

subdirectory ASCII under the actual project directory. Each line of the file must contain the following parameters:

“channel number” “offset” “lateral offset”

Channel number defines the channel number of a multichannel device (see above).

The smooth direction and the smooth size are taken from the values defined within the traceheader menu (update option “correct for offset in profile direction”).

Apply processing flow: if activated the processing flow named `importprocessingflow.pfl` if existing under the actual project directory will be automatically applied during the import. For the processing option manual gain (y) the file `ygain.import` (must exist under the project directory) will be used. If this file does not exist the processing option will be ignored as well as all the other processing options which require a parameter file for the sequence processing (e.g. static correction or muting).

1.5.7 Import time/comment specification

This group box allows the input of the time dimension, a comment and if necessary (if not available in the file header of the raw data) of the time increment and the sample number.

TimeDimension: Specification of time dimension. Possible inputs are ms and ns. The standard setting complies with the reading-format. If you select RADAN, RADAN_ANALOG, OYO8BIT, GEOTEL, TRS2000, PULSEKKO, MALA RD*, UTSI, GEOSCAN32, ASCII-PULSEKKO, ASCII-OYO8BIT, SEG2-RADAR or ASCII-MATRIX as reading-format ns is assumed as standard setting, for all others ms.

Comment: Giving an arbitrary comment which is saved in the header and can be recalled at any time. The comment is read from the raw data header and can be changed within the edit fileheader menu (fileheader input1).

time increment: Specification necessary for GEOTEL, OYO8BIT, ASCII-OYO8BIT, TRS2000, ASCII-MATRIX, FREEXXBIT or BITMAP data - specification of increment in time direction. Together with the number of samples (see below) the recording length per trace in the specified dimension is determined by the time-increment.

Time resampling: only for SEG2-data. The option allows to import only every n. sample. The option might be useful if the original number of samples exceeds the max. samplenumber of 65000. Of course the aliasing criterium must be fulfilled if a value greater than 1 will be entered.

sample number: Specification necessary for GEOTEL, OYO8BIT, FREE xxBIT or ASCII-OYO8BIT-data - specification of the number of digital samples per trace. Together with the time increment (see above) the recording length per trace in the specified dimension is determined by the sample number.

file header: specification of the size of the filheader in bytes. Specification is only necessary for FREE xxBIT. The information within this header will not be taken into account. The size must be entered in order to correctly read in the data.

trace header: specification of the size of the header of each trace in bytes. Specification is only necessary for FREE xxBIT data. The information within this header will not be taken into account. The size must be entered in order to correctly read in the data.

1.6 Export Menu

Use this option to export REFLEX formatted data into different data formats. The following formats are supported: BITMAP (BMP), SEG1, SEG2, GEOTEL, BGREMR, PULSEEKKO, MALA RD3- and RAMAC-BOREHOLE, NWF, RADAN and four ASCII-formats (ASCIISYNTH, ASCII-3COLUMNS, ASCII-MATRIX(TRACE/ROW) and ASCII-MATRIX(TRACE/COL)).

The option allows to export a number of 2D-lines. The filename of the single output name is equal to the filename of the original line plus an extension corresponding to the export format.

The option **export format** allows to specify the format of the data to be converted. At the moment the formats BITMAP (BMP), SEG1, SEG2, BGREMR, GEOTEL, PULSEEKKO, NWF, MALA RD3, RAMAC-BOREHOLE, RADAN-16Bit, RADAN-32BIT and five ASCII-formats (ASCII-3COLUMNS, ASCII-4COLUMNS, ASCII-5COLUMNS, ASCII-SYNTH, ASCII-MATRIX(TRACE/ROW) and ASCII-MATRIX(TRACE/COL)) are supported.

ASCII-3COLUMNS: each line contains the distance, time and the amplitude (format yy:xx where yy defines the number of places and xx defines the number of decimal places given within the global settings menu (chap. 1.2.1)). Depending on the settings within the AsciiColumns box the distance is either determined from the fileheader (**fileheader coordinates** activated) or set to the distance stored within the trace header (**traceheader coordinates** activated). Example for the use of the option traceheader coordinates: the data are nonequidistant and you want to write out the correct positions. With the option **header line** activated a header line will be inserted at the top of the ASCII-file. This might be of interest if the data shall be imported afterwards into Reflexw because the import of ASCII-3COLUMNS data needs a header line at the beginning of the file.

ASCII-4COLUMNS: each line contains the x-coordinate, y-coordinate, time and the amplitude (format yy:xx where yy defines the number of places and xx defines the number of decimal places given within the global settings menu (chap. 1.2.1)). Depending on the settings within the AsciiColumns box the x- and y-coordinates are either determined from the fileheader coordinates (**fileheader coordinates** activated) or they are set to the CMP-coordinates stored within the trace header (**traceheader coordinates** activated). Example for the use of the option traceheader coordinates: the data are nonequidistant and you want to write out the correct CMP-positions.

ASCII-5COLUMNS: each line contains the traceheader receiver x-coordinate, y-coordinate, z-coordinate, time and amplitude (format yy:xx where yy defines the number of places and xx defines the number of decimal places given within the global settings menu (chap. 1.2.1)). With the option depths or altitudes activated within the depths groupbox the converted depths or altitudes based on the given velocity are written out instead of the traveltimes. If altitudes are used the reference level is automatically taken from the actual z-coordinate. In both cases each line contains the x-, y-coordinates, the time, the converted depth (altitude) and the amplitude.

ASCII-MATRIX(TRACE/ROW): each line of the ASCII-file contains the amplitude values of one trace starting at trace 1 (row 1 = trace 1, row 2 = trace 2,). No header information is written out. The format for 32 bit data is given by yy:xx

where *yy* defines the number of places and *xx* defines the number of decimal places given within the global settings menu (chap. 1.2.1).

ASCII-MATRIX(TRACE/COL): each column of the ASCII-file contains the amplitude values of one trace starting at trace 1 (column 1 = trace 1, column 2 = trace 2,). No header information is written out. The format for 32 bit data is given by *yy:xx* where *yy* defines the number of places and *xx* defines the number of decimal places given within the global settings menu (chap. 1.2.1).

ASCII-GEOSOFT_XYZ: each line contains the distance, time and the amplitude (format 12:3). Depending on the settings within the AsciiColumns box the distance is either determined from the fileheader (**fileheader coordinates** activated) or set to the distance stored within the trace header (**traceheader coordinates** activated). Each scan in x-direction is separated from the previous one by a command line: "line xxx". The parameters **XSCALE** and **YSCALE** define a scaling value for the x-values and the y-values respectively. The parameter **threshold** defines a threshold value. Data values (absolute value) below this threshold are not written out.

ASCII-GRD: this format corresponds to the ASCII-GRD format used by the program SURFER. The radiobox **first sorting** defines how the data are written out onto the ASCII-file. With **y-(time)-direction** activated the data are written out trace after trace. Then the second line contains the number of samples and of traces. The third line contains the start and end-position in time-direction, the fourth line the start and end-position in profile direction. Each of the following lines contains all amplitude values of one trace. With **x-(distance)-direction** activated the data are written out sample after sample. Then the second line contains the number of traces and of samples. The third line contains the start and end-position in profile-direction, the fourth line the start and end-position in time direction. Each of the following lines contains all amplitude values of one sample.

To be considered for timeslices: in order to get the correct sorting within Surfer it might be necessary to use the x(distance)-direction for the first sorting.

To be considered for tomographic models: as the 0/0 position normally lies on the bottom/left within Surfer it might be useful to flip the y-axis (processing option flip-y-axis) and to use the y-axis as an elevation axis (enter the reference level within the fileheader - option starttime).

ASCII-3D_DATA: this format should be used for 3D-data (see also 3D-datainterpretation, chap. 3). Each line contains the x-, y-coordinates and the time and the amplitude. An exponential format is used. With the option depths or altitudes activated within the depths groupbox the converted depths or altitudes based on the given velocity are written out in addition. The ref.level will be taken into account if altitudes are used (altitude: ref.level-depth). Then each line contains the x-, y-coordinates, the time, the converted depth (altitude) and the amplitude.

SEGY and SEG-Y-DOS: SEG-Y uses swapped bytes, e.g. for an Unix maching, SEG-Y-DOS uses unswapped bytes (DOS, Windows format).

With the option **ps timeincr.** activated the segy-timeincrement within the exported data is assumed to be in picosec if the original timeincrement is in ns. The parameter **scaling factor** is used for scaling the distances and the xy-coordinates before export. The option is only necessary for the export format SEG-Y. The following bytes are written out on the traceheaders:

001-004: trace sequence number

009-012: original field record number

029-030: trace id code (1 = seismic)
 037-040: distance from source to receiver
 041-044: receiver group elevation
 045-048: surface elevation at source
 069-070: scaler for elevations and depths
 071-072: scaler for coordinates and for the distance
 073-076: Source x-coordinate
 077-080: Source y-coordinate
 081-084: group x-coordinate
 085-088: group y-coordinate
 089-090: coordinate units (1=length)
 115-116: Number of samples in this trace
 117-118: sample interval in μ s
 159-160: day of year
 161-162: hour of day
 163-164: minute of hour
 165-166: second of minute

With the option **load and save pick values** activated the picks stored under the same filename (option automatic name within the save pick menu) will be saved within the SEGY traceheader bytes 237-240 in next smaller timedimension (μ s if original timedimension is ms or ps if original timedimension is ns).

With the option **load header(s) from original SEGY-file** activated an original SEGY-file will be queried from which the file- and traceheader will be taken over. Only the sample number and the dataformat of the datavalues will be updated.

The program automatically controls whether the data are swapped or not independently from the choice SEGY or SEGY-DOS.

SEG2: non default header word Y_AXIS specifying TIME or DEPTH depending on the timedimension.

BITMAP: The following bitmap formats are supported: bmp, jpg, tif and pcx. The parameters **AutoSize**, **XSize** and **YSize** determine the size of the bitmap. With AutoSize activated the x- and y-sizes of the bitmap are automatically determined from the number of scans and samples. With AutoSize not activated you must manually enter the x- and y-sizes. If the entered x- or y-size number exceeds the trace or samplenummer an automatic interpolation in the corresponding direction is done for the pointmode presupposed the option **autointerpolation** is active. The max. x- or y-size in pixels is 30000. No flipping and rotation of the axis (see plotoptions) are supported.

If the depthaxis under Plotsuboptions (chap. 1.7.6) is activated or if the y-axis already is a depth axis the option **scale relationship** allows you to enter a scale for the depth and therefore also the timeaxis. In this case the y-size is determined from the x-size and this scale value. For example a number of 1 for the scale means that the x- and the depthaxis have the same scale. Example: if the x-axis of a file consisting of 2000 traces is 100 m long and the depth-axis is 10 m long (depth-axis/x-axis = 1/10), the bmp-file will have about 2000 x 200 pixels if the scale relation is set to 1. If the scale relation is set to 2, the number of pixels for the depth-axis will double and the bmp-file will have 2000 x 400 pixels, approximately. The option is available both with activated and deactivated AutoSize option.

If axis within the PlotOptions is activated, the axis in x- and y-direction are automatically added to the bitmap using the current font settings.

If a free datarange is present (e.g. when using the plotoption manual scaling) the 0 amplitude value color is used by default. If the option White background for free datarange within the global settings is activated the background color given within the plotoptions is used instead.

With the option **show name** activated the filename (without path) is shown at the top using the textfont.

With the option **showpicks** activated it is possible to display picks stored within individual pickfiles. After having pressed button start you must choose the wanted pickfiles from the filelist. Within the fileheader of each pickfile the corresponding datafilename is stored (see also Pick save Menuitem). This filename serves for the assignment of the chosen pickfiles to the chosen 2D-datafiles. To be considered: the datafilename size must not exceed 20 characters including file extension.

With the option **georeferenced tfw** activated the bitmap export is done together with a tfw world file. A world file file is a plain ASCII text file consisting of six values separated by newlines. The format is:

pixel X size

rotation about the Y axis (usually 0.0)

rotation about the X axis (usually 0.0)

negative pixel Y size

X coordinate of upper left pixel center

Y coordinate of upper left pixel center

The following plotoptions are automatically used:

no axis (plotoption showaxis automatically deactivated)

max. y-value on top (plotoption flip y-axis automatically activated)

RADAN 16bit, PULSEKKO, MALA RD3: The option **scaling** allows to enter a scaling value for the data. This might be useful in order to rescale data with a higher data representation than 16 bit.

PULSEKKO: with the option **Ekko_Project format** activated the hd file will be saved using the newer Ekko_project format.

Filefilter allows to enter a filter for the multi file choice.

Filepath specifies the path for loading the files to be processed.

addition for output filename for the automatic filenames: enter an additional specification for the outputfilename which will be added at the end.

manual outputfilename: if activated the output filename will be manually queried for each file to be exported.

generate ASCII report: if activated an ASCII report will be created containing the filename of the exported file, the start and end coordinates in profile direction and the time start and end coordinates as well as the number of traces and of the samples (points in x- and y-direction for the bitmap export).

Start starts the export. After having chosen the option, you must select the wanted files to be exported. The converted files will have the same name as the original file except the extension and will be stored under the path ASCII under the project directory.

1.7 PlotOptions

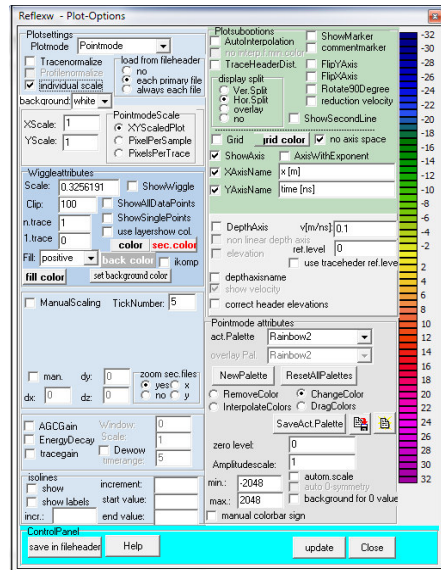
Within this menu you may change the current plot settings.

1.7.1 Plotsettings

Plotmode: The option allows the specification of the plottype. You may choose between pointmode and wigglemode.

Pointmode means that the data are plotted with colored pixels according to the predefined color-amplitude configuration (see also pointmodeattributes).

Wigglemode means that each trace of the profile is plotted as a polygonal line. The size of the deflections of each wiggle is controlled by the parameter scale (see also Wiggleattributes). The two modes might be used together if the pointmode is chosen together with the activated option **ShowWiggle**.



Three different scale modes are incorporated:

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

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

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

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

YScale: enter a value for the window scaling in y-(normally time-)direction. If PixelPerSample is activated, the parameter gives the range between successive

points in pixels. If PixelsPerTrace is activated, the parameter has no meaning. If XYScaledPlot is activated, the parameter gives the y-zooming value corresponding to the current profile window (e.g. a value of 2 means that half of each trace length is plotted into the current window).

Tracenormalize: Activate this option if you wish the data to be plotted amplitude normalized for each trace. With the option profilenormalize deactivated (see below) the maximum amplitude of each visible trace is normalized to 1. Like that a plotting is guaranteed, where all traces are well visible. Deactivate this option if you want to plot the data with real amplitudes. For the **wigglemode** the size of the individual deflections is controlled by the parameter Scale (tracenormalize: Scale = Size in Pixels, no tracenormalize: current amplitude * Scale = Size in Pixels). For the **pointmode** the parameter Amplitude scale controls the amplitude color assignment.

Profilenormalize: with this option activated the normalization is not done based on the max. amplitudevalue of each trace but on the mean amplitudevalue of the complete profile. Thereby amplitude variations from trace to trace within one profile will remain but it is possible to compare profiles with different value scales. The option is only available with the option tracenormalize activated. To be considered for

Wigglemode: The entered wiggle size corresponds to the mean amplitude. Therefore you must enter a clip value greater than 100 for the greater amplitudes. Otherwise these amplitudes will be clipped. By default a clip value of 200 is set.

Pointmode: by default the amplitudescale is set to 0.5 - this means that the color amplitude assignment includes all amplitude values until twice the mean amplitude. Higher amplitude values are assigned to the max. color(s).

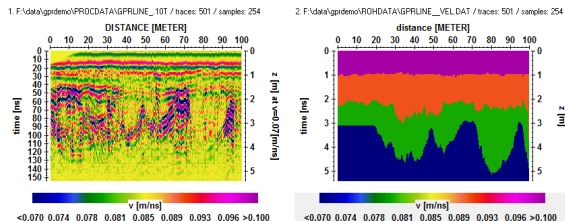
load from fileheader: controls if the plotoptions stored within the fileheader of each profile (plotmode, wigglemode attributes and pointmode attributes) are used.

With the suboption **no** activated the plotoptions stored within the fileheader will not be used. The current plotoptions remain unchanged even if a new primary profile will be loaded or if the projectdirectory will be changed.

With the suboption **each primary profile** activated the plotoptions are always updated from the fileheader of the primary profile after a new primary profile has been loaded. These plotoptions are used for all profiles to be displayed.

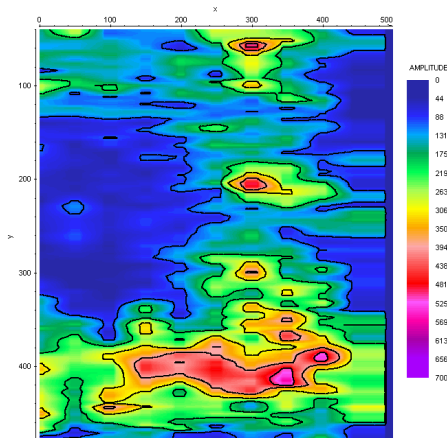
With the suboption **always each file** activated the plotoptions stored within the fileheader of each profile are always used for the display. This might be useful for example if profiles with very different amplitudes (e.g. a "normal profile" and the velocity distribution) shall be displayed

simultaneously or if profiles shall be displayed both using the pointmode and the wigglemode. If color bar display is active different color bars will be shown.



individual scale: only for the 2D-dataanalysis. If activated new pointmode scaling factors for the secondary windows may be used by entering a percentage value. The options for the scaling factors are located within the upper left corner of each secondary profile. The stepsize for the factors is 1 % unless the small option right to the scaling factor within the second window is activated. Then the stepsize is 10 %. The option may be useful if profiles exhibit different energy distributions.

Background: allows to set the background color for the 2D-dataanalysis and the 3D-datainterpretation. Four predefined colors are available - white, green, blue and grey.



Manual scaling: Activate this option if you wish to manually input the min. and max. axis lengths in x- and y- (time-) direction (see also chap. 1.7.4 Manual scaling and incrementation). Deactivate this option if you wish the program to automatically determine the axis lengths.

Man.: if activated the manual subdivision of the axis (options dx, dy and dz, see also chap. 1.7.4 Manual scaling and incrementation) will be enabled - if deactivated the program automatically chooses a suitable incrementation.

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

zoom sec. lines: the zoom for the secondary lines can be deactivated using the following suboptions no, x or y. With **no** activated no zoom will be done for both axis, with **x** activated no zoom will be done for the x(distance)-axis and with **y** activated no zoom will be done for the y(time)-axis.

Isolines: This option allows to plot isolines in addition (option **show** active). The option is only available for the pointmode. The **increment** together with min. value and max. value of the amplitude range of the pointmode attributes (see also chap. 1.7.2) control the number of the isolines:

1. isoline: min.value+increment
2. isoline: min.value+2*increment

...

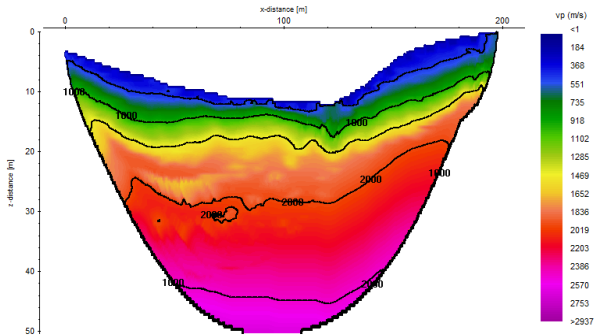
last isoline: max.value-increment

The color and the line thickness of the isolines are controlled by the settings of the symbol font.

The option is available within the 2D-dataanalysis, the 3D-datainterpretation and the modelling module. The CPU-time for plotting significantly increases if the option will be activated.

For the example on the right an increment of 100 and bold black symbol font was used. The min./max amplitude values were fixed to 0 and 700 respectively.

It is possible to enter different **start** and **end values** for the display of the isolines. In addition the individual isolines may be labelled (option **show labels** activated). The option **incr.** allows to specify a different increment for the display of the isoline which shall be labelled in addition. This value must fit the single isoline values which are defined by the start value and the isoline increment, e.g. incr. set to 1000, start value set to 500, end value to 2500 and increment set to 500.



1.7.2 Pointmodeattributes

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

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

overlay Pal.: load the wanted color palette for overlaying a second profile. In order to overlay the 2 profiles you must deactivate the options hor. and ver.split and activate the option ShowSecondLine. To be considered: by default the option ShowSecondLine is automatically deactivated when deactivating both the options hor. and ver.split for the pointmode.

One example might be overlying a profile with the underground velocity distribution e.g. taken from a seismic refraction analysis. In this case it might be useful to activate the plotoption always each file because of the different amplitude distribution.

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

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

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

Edit act. Palettes: enters the enter the color palettes editor menu which offers an additional detailed possibility to edit the color palettes - see chap. 1.7.2.1.



loads a new **palette file** from disk



saves the actual palette file on disk using any name.

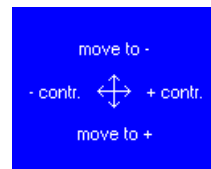
RemoveColor: with this option activated you may remove any color from the color bar on the right-hand side by clicking on the corresponding color using the left mouse key.

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

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

DragColor: two possibilities to interactively drag the colors are given

- a blue panel opens (see picture on the right) which allows to interactively change the colors when moving the mouse with pressed left mouse key within the panel. Starting from the midpoint increases the contrast to the right and decreases it to the left. The color table is moved to the colors which are assigned to negative amplitude values when moving to the top and vice versa when moving to the bottom. The changes are instantaneously shown within the loaded profile.



- you may continuously move a linearly interpolated color range. After having activated this option, please click on two different colors. In-between these colors a linear interpolation is done. The trackbar below may be used in order to increase or to decrease this color range.

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

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

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

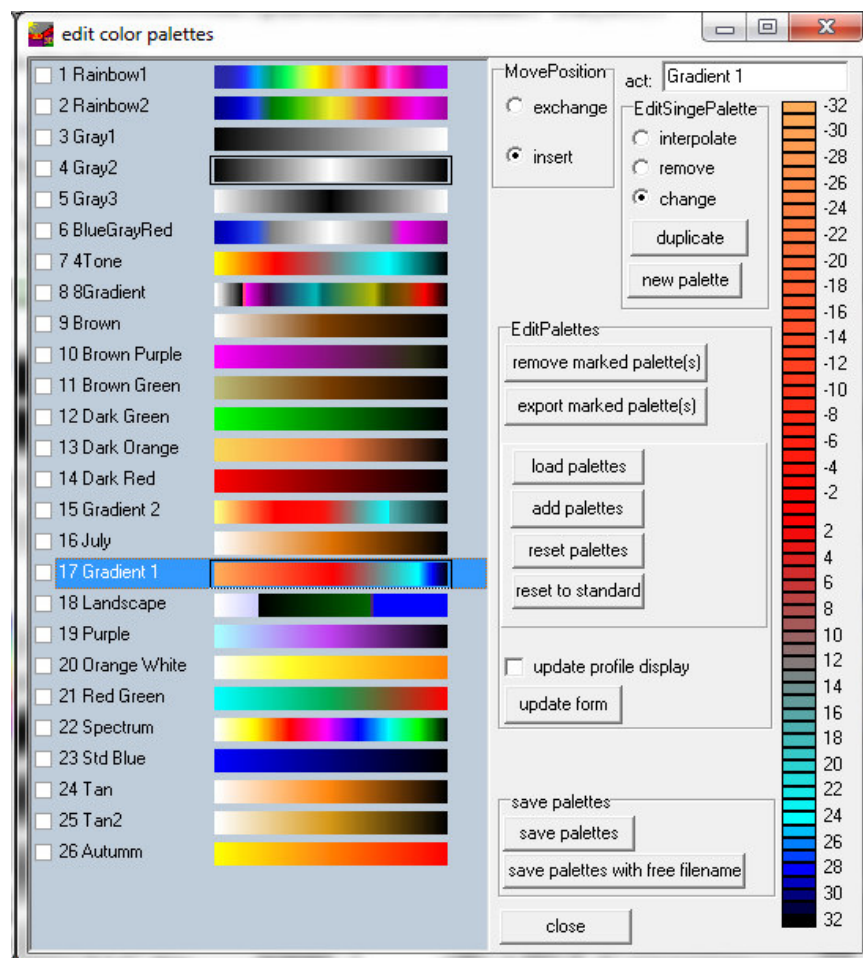
autom.scale: the option allows to automatically define the min./max. amplitude range for the color/amplitude assignment. If activated the program extracts the min. and max. amplitudes from the actual profile or slice and uses these values for setting the color palette. The amplitude scale cannot be changed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

background for 0 value: If activated all zero values will be plotted using the background color.

Manual colorbar sign: if activated a manual name for the colorbar sign may be entered. Activate this option if the sign does not fit correctly.

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

1.7.2.1 edit color palettes



When opening the edit color palettes menu the actual palettes are displayed within the left panel. Here you can activate one palette by clicking on it. With the option **update profile display** the actually loaded line will be displayed using the new palette. Activating the option **update form** updates the display of the edit color palettes menu.

The **order** of the palettes may be simply modified by tracking one palette with pressed mouse key. The option **MovPosition** controls therepositioning. With **insert** activated the palette will be positioned at the new place and all other palettes will be moved accordingly. With **exchange** activated the position of only two palettes will be changed. The activated one will be placed at the new position

whereby the exchanged palette will be placed at the old position of the actual palette.

Within the **EditSinglePalette** groupbox the actual palette may be changed.

Duplicate allows to duplicate the palette in order to make changes afterwards.

The option **new palette** allows to create a completely new palette. Choose 16 different colors from the color dialogue menu. Between these 16 colors a linear interpolation is automatically done in order to define the 128 different colors.

After having entered the colors the name of the palette is queried and the palette will be attached at the end.

The three options **interpolate**, **remove** and **change** allow to edit the actual palette.

With **interpolate** activated you may interpolate the colors between two choosable colors. Click on the two colors in the color bar using the left mouse key. After having chosen the second color, the in-between colors are linearly interpolated. With the option **remove** activated you may remove any color by clicking on the corresponding color using the left mouse key. With the option **change** activated you may change any color by clicking on the corresponding color using the left mouse key. The new color must be defined in the opened color dialogue menu.

The groupbox **EditPalette** may be used for editing a complete color palette group. The individual color palettes may be marked within the field at the right side of each palette. The marked palettes may be removed (option **remove marker palette(s)**) - or exported to a new file - option **export marked palettes(s)**).

The option **load palettes** allows to read the palettes from any existing palette file (extension *.fil). All existing palettes will be removed.

The option **add palettes** allows to add palettes from any existing palette file (extension *.fil). These new palettes will be placed behind the last actual palette.

The option **reset palettes** removes all existing palettes and reads the palettes from the file palette.fil.

The option **reset to standard** only keeps the internal standard palettes and removes all other ones.

Within the **save palettes** groupbox the actual palettes may be saved under the filename palette.fil (option **save palettes**) or under a free filename (option **save palettes with free filename**).

1.7.3 Wiggleattributes

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

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

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

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

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

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

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

Sec.Color: choose the wiggle color for the secondary line if primary and secondary line are plotted into one image (no horizontal or vertical split).

Back Color: choose the color for the wiggle background.

set background color: sets the wiggle background color to the overall background color.

Use layershow colors: if activated the layershow colors are used for plotting and filling the wiggles. Each trace has a different color (max. 100). The layershow colors can be defined within the layershow menu (see chap. 1.12.3, option colors).

Fill: controls the filling of the polygonal lines.

Entering **no**, means no filling.

Entering **positive**, means only the positive amplitudes are filled with the chosen FillColor.

negative means only the negative amplitudes are filled with the chosen FillColor.

colors means that the wiggles (positive and negative amplitudes) are filled with the current color amplitude assignment. In this representation correlated signal arrivals are very well recognizable.

Pos./neg. means that both the positive and negative wiggles are filled using the current fill color.

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

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

ikomp: if activated the secondary color will be used for traces with ikomp = 2.

1.7.4 Manual scaling and incrementation

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

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

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

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

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

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

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

1.7.5 PlotGain/Filter

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

AGCGain: if activated an AGC (AutomaticGainControl) with the given **Window** value (enter the wanted window length in the current timedimension) is applied for plotting the data. AGC facilitates the creation of equally distributed amplitudes in y-direction (normally time axis) within a predefinable window. This option serves for emphasizing of low amplitude ranges against ranges with high

amplitudes. The information of the true amplitude is lost, of course. The program calculates at first an average amplitude over the total time range for each trace. After that the program scales each amplitude value in that way that the mean amplitude has the same value for each selected window around the current value with in a trace. The size of the window determines the kind of amplitude equality distribution. A window size of only one sample increment means that each time sample within one trace receives the same amplitude value (no reasonable choice), a window size of the whole trace length causes no modification of the amplitude. Small window sizes cause a strong equality distribution, large windows a weak. Often it is necessary to apply a scaling factor (parameter **Scale** with which the data are multiplied after the application of the AGCGain) smaller than one because after the application of the AGCGain some amplitude values will exceed the maximum amplitude of the original profile. The optimal choice of this scaling factor enables that all amplitude values will not exceed the limit of 16 bit.

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

tracegain: if activated the gain factors stored within the traceheader are used for the display. The option is only available for data stored with the new 16 bit integer or new 32 bit floating point format (see also Import format specification, chap. 1.5.4). The option has no effect with activated option `tracenormalize`.

Dewow: With this option activated a running mean value is calculated for each value of each trace. This running mean is subtracted from the central point. As filter parameter the **timerange** for the calculation of the running mean value must be entered. This filter may be used for eliminating a possible low frequency part (dewow). For this purpose the window range should be set to about one principal period. This filter has been included within the `plotoptions` in order to be able the raw MALA RD3 or PULSEEKKO data without applying any processing step. In any case you should use the processing step `subtract-mean(dewow)` processing step (chap. 1.11.1.9) for the "normal" interpretation because other filters and interpretations tools are affected by a non dewowed datafile in a non desired way.

1.7.6 Plotsuboptions

This group box controls the main plot settings.

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

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

TraceHeaderDistancies: if activated the profile is plotted based on the individual distances stored in the single traceheaders and not based on the equal trace increment of the fileheader. The option is available both for the wiggle and point plotmode. With activated option the following restrictions for the pointmode hold true:

- Printing is not possible
- the actual setting of the parameter n.trace is also valid for the pointmode
- for the filling a mean fill size will be determined automatically. In case of strong variations of the trace positions gaps may occur. The same holds true for the scrolling. Please use after scrolling the repaint button.

The actual setting of the TraceHeaderDistancies will be kept after leaving the program if the wigglemode has been chosen. In case of the pointmode the default deactivated setting will be used.

A repositioning of the data into equidistant traces instead of the TraceHeaderDistancies is recommended if you are using filters which require equidistant data (e.g. migration or fk-filter). The processing step "make equidist.traces" (chap. 1.11.6.3) under processing/trace interpolation performs this repositioning.

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

no interp.f.min.color: if activated no interpolation will be done for amplitude values lying within the min. color range. This might be useful for example for plotting a tomographic inversion result which has been restricted to the covered area. Then the border of the area will not be interpolated. The option is only enabled if AutoInterpolation has been activated.

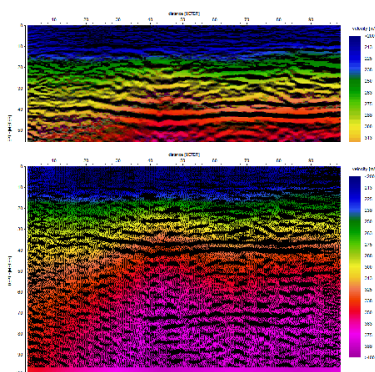
display split: this group box allows different display split possibilities (option is not available for REFLEXW DataView):

Ver.Split: if activated the screen has a vertical split, this mean the display is splitted vertically.

Hor.Split: if activated the screen is split horizontally (option is not available for REFLEXW DataView).

- If only two profiles have been loaded:

Overlay: the two loaded profiles will be overlaid within one single window. The files should have the same scale in distance (x) and time (y) direction. Apart from comparing files of identical type (e.g. different filtered lines) another example might be overlying a profile with the underground velocity distribution e.g. taken from a seismic refraction analysis. Different combinations of point and wiggle mode together with the load from fileheader option are supported:



Pointmode for both profiles: the 2 profiles will be overlaid using the color palettes defined within the option "act palette" (1. line) and "overlay Pal." (2. line). If the option "load from fileheader" is set to "always each file" different plotscales are used. In this case the plotscale for the primary file still can be changed manually. It is recommended to use a grayscale for one of the profiles because otherwise the colours will be changed significantly by the overlay.

Combination of point and wiggle mode: this is possible if the option "load from fileheader" is set to "always each file". The fileheader of one profile must contain wiggle mode and the fileheader of the other profile must contain pointmode. The plotscale for the primary file still can be changed manually.

Wigglemode for both profiles: If the plotmode is set to wigglemode two different wiggle colors are used for the different profiles.

No: the option "ShowSecondLine" will be deactivated and only one profile without split will be plotted.

- If more than 2 profiles have been loaded (see option "open 1.-4.line")

ver./hor.split: the data windows are both splitted vertically (first) and horizontally.

hor./ver.split: the data windows are both splitted horizontally (first) and vertically. The different split options now can also be reached directly via speed options within the 2D-dataanalysis menu.

ShowSecondLine: if activated the second line is plotted (option is not available for REFLEXW DataView).

Rotate90Degree: if activated the profile is rotated by 90 degrees. With the option **elevation** activated the distance(depth) axis is replaced by an elevation axis showing the elevations based on the entered **ref. level**.

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

Only slices: only visible for the 3D-datainterpretation - if activated the activated FlipYAxis option only acts on the slices.

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

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

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

no axis space: if activated no spaces will be used for the axis display.

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

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

AxisWithExponent: if activated the axis labelling is done with exponential representation if the labelling values exceed some predefined values (e.g. values between 10000 and 30000 are displayed as $100 \cdot 10^{**2}$ and $300 \cdot 10^{**2}$).

If activated and no manual axis labelling (options XaxisName and YaxisName deactivated) is entered the following holds true in addition:

timeaxis: automatic display in μ s instead of ns and ms instead of μ s and s instead of ms if the timerange is bigger than 10000.

distanceaxis: automatic display in KM instead of METER if the distancerange is bigger than 10000.

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

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

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

non linear depth axis: if activated the depth axis is calculated from the timeaxis and a 1D-depth velocity distribution which is taken from a previously stored 1D-velocity file (see CMP-velocity analysis menu, chap. 2). When activating this option the filename of the wanted 1D-velocity file is queried. The option is only valid if the options flip y-axis and Rotate90Degree are disabled.

Elevation: With the option elevation activated the depth axis on the right hand side is replaced by an elevation axis showing the elevations based on the entered **ref. level**. With the option **use traceheader ref.level** activated the reference level will be taken from the maximum value of the traceheader rec. z-coordinates and does not need to be entered manually. The current elevation is calculated from: reference level - current depth value.

If the option **Rotate90Degree** is activated the distance(depth) axis is replaced by an elevation axis showing the elevations based on the entered **ref. level**. In this case the optional right hand side depth axis will not be affected.

Only right axis: option only enabled if Elevation has been activated. If the option is activated, in case of a distance/depth section (e.g. after a time depth conversion) the elevation axis is drawn on the right side only. The left axis will be drawn as a depth axis.

correct header elevations: if activated the traces are shifted based on the receiver and the shot elevation values stored within the traceheader of each trace and the entered elevation level. The shift levels are calculated from the difference of the entered elevation level and the individual traceheader elevation values. Based on the current velocity the travelttime shift value is calculated from the sum of the shot and the receiver elevation differences.

The correct header elevations plot option is comparable with the processing option correct3Dtopography in the processing menu StaticCorrection/muting, if the static correction is done based on the shot and receiver elevations stored within the traceheaders. The time range will be automatically adjusted corresponding to the maximum shift level.

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

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

1.7.7 ControlPanel

save in fileheader: saves the actual plotsettings within the fileheader of the actual profile.

Update: allows to show the actual profile with the new plotoptions without leaving the plotoptions menu.

1.8 Print Menu

This menu allows you to setup the parameters for printing a profile. In every case the complete profile is printed according to the chosen minimum and maximum border values (either automatically determined or entered manually - see also manual scaling). The actual picks will also be printed if the pick panel is active.

Batch printing is supported - see PrintOptions2 (option several files).

The scale in x- and y-direction is freely choosable (see also PrinterSize). The scale is entered either by direct input of the scale or by entering the length of the individual x- and y-axis.

A page blocking option is included (see also PrinterSize and PrintOptions1).

Printing on banner paper (printing on continuous paper) is supported.

If printing a layershow you may choose between printing both the profile and the interpretation (options **print profile** and **print layershow** activated) and only the profile (option **print profile** activated) or the interpretation (only option **print layershow** activated).

If a free datarange is present (e.g. when using the plotoption manual scaling) the 0 amplitude value color is used by default. If the option White background for free datarange within the global settings is activated the background color given within the plotoptions is used instead.

1.8.1 PrintOptions1

This group box allows to specify some controlling options for the scale and the print view.

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

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

Y-scale output: the given scale is used for determining the y-size of the output. With layer-show activated (see also LayerShow MenuItem) the scale is applied on the depth axis of the layer show in the lower image and not on the time axis of the 2D-profile in the upper image.

With depth axis activated (see also Plotsuboptions) the scale is applied on the depth axis and not on the time axis of the profile.

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

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

use current zoom: if activated the currently set zoom parameters are used for the printing. If deactivated the complete profile(s) is plotted. The option is only enabled for printing from the 2D-dataanalysis.

1.8.2 PrinterSize

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

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

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

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

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

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

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

get total print size: allows to show the total print size in x- and y-direction.

1.8.3 PrintOptions2

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

filename for title: if activated the actual filename will be used for the printer title and the default filename for a pdf output. The file extension will be kept (the '.' is replaced by '_').

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

print file comment: print the comment stored in the file header of the profile to be printed. This comment is printed at the lower left corner of each line to be printed. When printing the Windowing 3D-file the file comment is only printed when a 3D-file has been loaded.

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

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

print several files: activate this option if you want to print out several files with the same printer attributes and printer lengths. The current plot settings are valid. When printing from the menu **2d-dataanalysis**, a filechoice query appears where you must select the wanted profiles. All chosen profiles are printed on separate paper sheets. With the option **autom.** load within the pick panel the picks saved under the same file name as the selected profiles are automatically loaded and printed together with the profile.

When printing a **3D-file** from the menu **3D-datainterpretation** with the option **scroll** activated all current slices are printed on separate paper sheets.

When printing **several 2D-lines** from the menu **3D-datainterpretation** with the option **scroll** activated all currently chosen profiles are printed on separate paper sheets.

When printing **several 2D-lines** from the menu **3D-datainterpretation** with the option **window** activated a filechoice query appears where you must select the wanted profiles. The chosen profiles are sorted alphabetically and are printed according to the current window combination. An example shall make the proceeding clear once more: You want to print 20 parallel profiles and for this combine 4 profiles each time in 2*2 windows. First you have to choose 4 arbitrary sample profiles and make the corresponding window selection. In the next step you choose the print button with the option **print several files** activated and select the desired profiles. The program then automatically prints the first 4 profiles on one paper sheet. After terminating the output the next 4 profiles are taken and processed in the same way.

print header boxes: if activated the program asks for the header box file to be loaded after having activated the option **print**. For a detailed description of the header comment boxes see chap. 1.8.7 (print preview).

1.8.4 FontSettings

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

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

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

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

control fontsize: active only when printing within the windows mode of the **3D-datainterpretation**. Deactivate this option if the fontsize is not correctly determined from the program. In this case the screen fontsize is used for printing. Therefore you must change the fontsize in such a way that the printer and the screen resolution match.

1.8.5 ControlPanel

PrinterSetup: Enter the printer-setup menu.

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

Bitmap size: this is an additional check for the used printer bitmap. Specify the maximum number of pixels (x-pixels multiplied with y-pixels) within each bitmap that will be used to print the profile in point mode. If the number of pixels of the whole bitmap exceeds this value, the program will automatically perform partitioning and use smaller parts of the bitmap for the whole bitmap. Again if problems occur with the printing (e.g. something is missing or not clearly displayed) it might be helpful to decrease this number. The number does not restrict the total number to be printed.

Preview: Enter the print preview menu which allows a preview of the size of the printoutput and to define text boxes (for a detailed description see chap. 1.8.7).

CANCEL: break off printing

PRINT: start printing

1.8.6 Printing on banner paper

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

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

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

1.8.7 Print preview

The print preview window is only available within the 2D-dataanalysis.

The print preview menu allows you to preview the size and shape of the print output and to define a **print header** consisting of up to 30 different header comment boxes containing up to 6 different comments. The print area is whitened and a simple preview of the profile(s) is shown within this area.

To be considered: The profile preview is only a simple bitmap transform from the screen and does not exactly represent the print output especially concerning the axis size. Some print options like print general comment or print file comment and print frame are also not taken into account.

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

The following general options are available:



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



replot with current zoom parameters



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

close: close the print preview window without printing.

print: does printing using the current comment box settings and closes the print preview window.

reset act: clear the current comment box.

reset all: clears all comment boxes.

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

save: saves the existing set of comment boxes on file. The file will have the extension hea and is stored under the path rohdata under the current project directory. You must store the set of comment boxes if you want to use the boxes without entering the print preview window (see option print header boxes within the PrintOptions2, chap. 1.8.3).

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

copy to all pages: copies the actual print header to each page. The option may be used if the same or nearly same print header should be used for each print page.

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

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

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

X-pos: enter the x-coordinate of the left corner in cm

Y-pos: enter the y-coordinate of the top corner in cm

width: width of the box in x-direction in cm

height: height of the box in y-direction in cm

degree°: specifies the rotation angle of the comment text - e.g. 0°: horizontal alignment, 90°: vertical alignment

pen width: specifies the width of the box frame

frame: specifies the color of the box frame

fill: specifies the fill color of the box if transparent if deactivated

transparent: if activated the box is transparent, activate this option for example if you want to place some comments within your data

bitmap-file: if activated a bitmapfile can be loaded into the box, e.g. a logo.

The **current box** is highlighted by a big frame. An existing box may be activated by pressing the left mouse button within the wanted box. With pressed left mouse button the current box may be moved to any position.

The edit option **box nr.** shows you the number of the current box. This option also allows you to choose any of the existing boxes.

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

Each comment box may contain up to **6 different comments**. Each comment is defined by the following parameters:

the comment text itself

font: enter the wanted font of the comment

X-Pos.: enter the x-position in cm within the comment box. You may use the up and down arrows for a fast replacement of the comment. The option change size defines the stepwise size of each redefinition.

Y-Pos.: enter the y-position in cm within the comment box. You may use the up and down arrows for a fast replacement of the comment. The option change size defines the stepwise size of each redefinition.

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

1.9 FileHeader Edit

Within this menu you may change the file header parameters of the current profile.

1.9.1 Fileheader coordinates

This group box allows the input of the coordinates stored in the fileheader (see also Fileheader-coordinates).

DistanceDimen.: input of the desired distance-dimension. Possible inputs are MM, CM, METER, INCH, FOOT or TRACENR.. Choosing TRACENR. means that the trace increment is automatically set to 1 and the name of the axis in profile direction is set to LOCATIONNUMBER.

ProfileDirection: enter the axis in which the profile is orientated. Possible Inputs are X, Y or Z.

ProfileConstant: enter the axis perpendicular to the profile orientation. Possible Inputs are X, Y or Z. With the setting of the ProfileDirection and the ProfileConst the so-called profile plane is determined.

XStart: enter the starting coordinate in x-direction.

XEnd: enter the ending coordinate in x-direction.

YStart: enter the starting coordinate in y-direction.

YEnd: enter the ending coordinate in y-direction.

ZStart: enter the starting coordinate in z-direction.

ZEnd: enter the ending coordinate in x-direction.

FixTraceincrement: With this option activated the trace increment of each file will remain fixed. This means when changing the starting coordinate in profile direction the corresponding ending coordinate will be automatically changed based on this new coordinate and the fixed trace increment. Also when changing the traceincrement only the endcoordinate will be changed. A manual change of the endcoordinate is not possible.

With this option deactivated the trace increment will be changed based on the set starting and ending coordinates.

Fix endcoordinate: This option is only available with active option FixTraceincrement. If activated the start coordinate will be changed instead of the endcoordinate when changing the traceincrement or the endcoordinate. A manual change of the startcoordinate is not possible in this case.

trace increment: enter the trace increment, i.e. the increment between successive traces. Input in Distancedimension (see above).

trace number: enter the number of traces for the current file.

line increment: only for 3D-file - enter the distance between parallel lines.

number: enter a number between 0 and 99 which is used for the automatic naming of the profile if the filename specification is set to automatic name.

utm zone: the utm zone which has been automatically used for the UTM-conversion will be displayed and may also be changed

1.9.2 Fileheader time specification

This group box allows the specification of the parameters of a single trace within with the profile.

TimeDimension: Specification of the time dimension. Possible inputs are ms and ns.

time increment: Specification of the time increment in the give time dimension. Together with the number of samples (see below) the recording length per trace in the specified dimension is determined by the time-increment.

sample number: specification of the number of digital samples per trace. Together with the time increment (see above) the recording length per trace in the specified dimension is determined by the sample number.

start time: specification of the start time for sample 1 in the given time dimension.

The total **time range** will be displayed below.

1.9.3 Fileheader input1

This group box allows the specification of some important fileheader parameters.

data type: There are seven different types: const. offset, single shot, several shots, 3D-const.offset, timeslice, single shot/boreholes and single shot/VSP.

The data type **'const. offset'** can be used for the import if the measurement is done along a line using a fixed offset between shot and receiver (e.g. Constant-Offset (CO) radar data).

If the data set is the result of a measurement with only one shot point but many receivers in a line, the use of the default data type **'single shot'** data is recommended (e.g. Common-Mid-Point (CMP) radar data, refraction seismic data). **Rec.start** and **rec.end** mean the start and end positions of the receivers along the profile line. **Lat.Offset** means the y-offset (perpendicular to the profile line) of the receivers. **Shot position** means the position of the shot along the profile line. **Shot lat.offset** means the y-offset

fileheader input1

data type: const.offset

S/R-distance: 0

Shot(CMP)-position: 0

shot lateral offset: 0

nominal frequency: 38.54527 get

comment:

acquisition date:

GainValues scans/s: 0

channel number: 0

value dimension: AMPLITUDE

(perpendicular to the profile line) of the shot. By default the shotx and recx positions within the traceheaders describe the location along the surface and the shoty and recy positions describe the y-offsets. A synchronization of the overall fileheader and the traceheader coordinates has been done during the import or can be done afterwards.

In case of data, which contain any kind of multiple shot / multiple receiver data, the default data type **several shots** should be used (e.g. reflection seismic - one data set combined of individual CMP data sets).

The data type **3D-const.offset** should be used for a 3D-data file consisting of several parallel 2D-lines (see also 3D-datainterpretation). This datatype should be used in combination with the conversion mode 3D-file equidistant or combine lines. The ensemble nr. within the REFLEXW traceheader controls the storing of the individual 2D-lines. All traces belonging to one 2D-line have the same ensemble-number which is stored within the REFLEXW traceheader (see chap. 1.9.6).

The data type **timeslice** should be used if the original data are timeslices i.e. time constant cuts within the XY-plane. This data type is also automatically used if the timeslices are created within REFLEXW (see chap. 3.6). The profiledirection should be X and the profileconstant Z. The profileconstant coordinate defines the position of the timeslice in time(z)-direction. Whereby the coordinates in x-direction are given corresponding to the standard const.offset type data, the coordinates in y-direction are defined like the parameters in time-direction for the const.offset data. Therefore the increment in y-direction is given by the y-increment which corresponds to the timeincrement for the const.offset type data and the start position in y-direction corresponds to the starttime for the const.offset type data and is taken from the original data if the option read starttime is activated. Otherwise it is set to 0 and may be entered afterwards within the fileheader menu.

The data type **single shot/boreholes** can be used for a borehole/borehole geometry. Offset means the offset of the receiver borehole along the surface. Rec.start and rec.end mean the start and end positions of the receivers within the borehole. Shot-offset means the offset of the shot borehole along the surface. Shot position means the position of the shot within the borehole. By default the shotx and recx positions within the traceheaders describe the location along the surface and the shoty and recy positions describe the locations within the boreholes. A synchronization of the overall fileheader and the traceheader coordinates has been done during the import or can be done afterwards. The conversion sequence inline lines may be used in order to update the shot position for different shots (line distance means the increment of the individual shots in this case). For a fixed receiver line and moving shots e.g. activate move shots and deactivate move receivers and enter the shot increment under line distance.

The data type **single shot/VSP** can be used for a borehole/surface geometry. The receivers are assumed to be placed within the borehole and the shot is placed at the surface. Offset means the offset of the receiver borehole along the surface. Rec.start and rec.end mean the start and end positions of the receivers within the borehole. Shot position means the position of the shot along the surface. By default the shotx and recx positions within the traceheaders describe the location along the surface and the shoty and recy positions describe the locations within the boreholes. A synchronization of the overall fileheader and the traceheader coordinates has been done during the import or can be done afterwards. The conversion sequence inline lines may be used in order to update the shot position for different shots (line distance means the increment of the

individual shots in this case). For a fixed receiver line and moving shots e.g. activate move shots and deactivate move receivers and enter the shot increment under line distance.

S/R-distance: enter the distance between source and receiver in the given distance dimension. This value is for example used for some CMP-based calculation like in the CMP-velocity menu.

shot position: enter the shot position within the CMP or moving line. By default the shot location is set to 0. For a correct velocity analysis this value has to be correctly specified.

nominal frequency: enter the nominal frequency of the antenna (GPR-data) or seismic source (seismic data) used. Enter the value in MHz with timedimension set to ns, KHz with timedimension set to μ s or Hz with timedimension set to ms. The option get allows to automatically determine the nominal frequency from the data.

Comment: Giving an arbitrary comment which is saved in the header and can be viewed or printed. 2 lines of comments with a maximum length of 120 characters for each line can be entered.

GainValues: This option allows the specification of the gain curve used for the acquisition of the data. The curve consists of up to 8 different gain values at individual time points. In-between these points a linear interpolation of the curve is automatically assumed. For some data this curve is automatically read in during the raw data conversion (e.g. from RADAN data - to be considered: restriction to max. 8 gainvalues - more gainvalues within the RADAN-file will be ignored). Using the option remove header gain within processing/gain (chap. 1.11.2.4) this gaincurve can be removed from the data.

value dimension: used for the description of the color bars.

If given the acquisition date, the scans/s and the channel number for a multichannel systems are also listed (cannot be changed).

1.9.4 Fileheader filename specification

specification: this option allows you to specify the naming of the converted profiles. Entering ***automatic-name*** means that the naming is based on the entered profile coordinates in profile constant direction (see below). Choosing ***manual input*** means that the filename must be entered manually. Choosing ***original name*** means that the original name of the raw data is automatically used.

filename: enter a name of the converted file provided that the specification is set to manual input.

Filenamefactor: used if the specification is set to automatic name. The start-coordinate in profile-constant direction is multiplied with that factor for naming. A

value larger than 1 (e.g. 10) is for example reasonable if floating point numbers should enter the naming. The current coordinate is of course not influenced by this value.

1.9.5 Fileheader ControlPanel

MarkerRelocate: allows the relocation of the start- and end-profile-direction coordinate based on the marker defined by the parameter **number** and the preset **position**. The program searches the number. marker within the line and sets its position to the preset position value. The start and end coordinates in profile direction are automatically updated (see also Edit several FileHeaders). With the option **comment marker** activated the comment markers are loaded in addition to the distance markers (see also edit comment markers).

ShowTraceHeader: shows the headers of the individual traces - see also trace header Edit (chap. 1.9.6).

update from traceheader: only active for the datatypes single shot, single shot/boreholes and single shot/VSP - allows to update the fileheader coordinates based on the traceheader coordinates. The option is useful in order to fit the fileheader and the traceheader coordinates



enter PlotOptions. Enter this option, if you want to change the plot parameters and to save them in the fileheader of the current file.

SAVE button: stores the changes in the fileheader of the current profile.

CANCEL button: leaves this menu without saving the changes.

1.9.6 Traceheader Edit

Within this menu you may show and edit (only enabled if the menu is called from the CMP-geometry menu) the trace header informations of each trace of the current profile.

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

distance: enter the distance along the profile

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

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

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

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

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

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

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

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

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

field record nr. (component): shows the field record nr. of the original data. This value may be used for defining a standard geometry within the CMP-processing (see also chap. 1.12.4.1). This value is read from SEG2 or SEGY original data if defined there. In the case of multicomponent data this value indicates the component (see also chap. 1.12.8).

scan direction: defines the scan direction of the individual traces of a constant offset or 3D-constant offs. file. During the import the scan direction is set to 1 for a profile direction of "X" and is set to 2 for a profile direction of "Y". The parameter scan direction is used for the generation of a 3D-datafile (type "use

independent interpolation scheme for pure x- and y-2D-lines”) based on traceheader coordinates.

time delay: specifies an individual time delay for each trace. The values defined here may be used within the static correction processing option.

Timecollect: shows the original time when recording the trace

original trace number: defines the original trace number within the original file. The number may differ from the Reflexw trace number, e.g. after an x-flipping.

SaveOnASCII: The option allows to save the traceheader coordinates on an ASCII-file using free positioning and separator. Activating this option opens a new window where you may define which parameters shall be written on the ASCII-file. Each line of the ASCII-file contains the desired informations of the individual trace. In addition to the parameters mentioned above two additional parameters can be written out: marker and TimeCollect. Marker values > 0 and < 100 define a distance marker, marker values >= 100 a comment marker. TimeCollect is

the acquisition time in seconds after midnight. This value may be used in order to synchronize time based GPS-data (see option **update** based on **GPS-times**). Each line contains the desired parameters for one trace in the order (place) the parameters have been chosen. The option reset resets the choice. The small buttons “s” and “g” allow to save and load the actual settings on the file TraceHeaderAsciiFormat.txt stored under the program directory. With **n.trace** greater 1 it is possible to restrict the number of traces to be written out.

Three different **separator** characters can be chosen (Tab, Comma or Blank). The option **start** stores the selected traceheader values of the actual file within the ASCII-file with the given filename.

The option **batch start into one file** allows to save the selected traceheader values of different files within one ASCII-file. The option **batch start** allows to

save the selected traceheader values of different files within different ASCII-files. The filenames of the ASCII-files are automatically determined from the data filenames. After having activated one of the two options the wanted files must be selected (multiple file choice using the ctrl or shift key).

With the option **dst fileformat** activated the dst fileformat is used for saving the traceheader parameters.

apply: applies the changed geometry on the current file (only enabled if the menu is called from the CMP-geometry menu).

update: The group box allows to update the geophone-, shot- and CMP-coordinates based on the distance coordinates of the file header (option **type** is set to **fileheader** - coordinates in profile direction and in profile constant - this is also possible within the Edit several FileHeaders menu, chap. 1.10; for a 3D-datafile you should use **fileheader 3D**) or based on a special circle geometry (option **type** is set to **circle**) or the coordinates are read from an ASCII file (option **type** is set to any of the different **ASCII-file** options).

To be considered for all tracenummer based gps-synchronizations like Ramac-GPS, IDS-GPS, GSSI-GPS, UTSI-GPS or PulseEkko GPS:

the original trace numbers are used for the synchronization of the GPS coordinates if a format was used in which the synchronization is based on the trace numbers. This allows processing steps that change the sorting or the number of original trace numbers (e.g. x-flipping in profile direction if the option meandering has been activated) to be carried out before the GPS coordinates are updated. With the option **use act trace no** activated the actual Reflexw tracenumbers are used instead of the original ones. This might be useful if the gps-datafile has been created afterwards based on the new tracenumbers after any processing step, e.g. a x-flipping.

If any of the following ASCII-files will be used for updating the traceheader coordinates a **gps marker** will be placed at all given positions if the global settings option **update from GPS ASCII-file** has been activated.

The option **fileheader** takes into account the start and end coordinates in profile direction and in profileconstant as well as the S/R distance for const.offset profiles. The S/R distance is added to the source coordinates in the given profile direction. This allows you to easily define the geometry for multi common offset data, e.g. GPR data with different antenna separations.

The options **interpolation** and **interpolation 0-data** and **interpolation equal data** allow the interpolation of the position data stored in the trace header, if for various traces no corresponding values exist.

The option **interpolation** uses a special flag within the traceheader of each trace which during the import of the data is set to 0 by default and set to a special value when traceheadercoordinates have been recognized. Only these traces where this flag is 0 are interpolated.

The option **interpolation 0-data** controls if all traceheadercoordinates of one trace are 0 and performs an interpolation for those traces. With the suboption **ignore time collect** activated the 0 data interpolation (extrapolation) will be also done for those traces for which a value for timecollect has been specified.

The option **interpolation equal data** controls if identical coordinates for the x- and y- shot and receiver positions for subsequent traces are present and performs an interpolation for those traces.

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

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

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

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

With the type **ASCII-file/interpol.** activated the data are read again from an ASCII-file (see type ASCII-file). In addition an automatic interpolation is done if for various traces no coordinate data are present within the ASCII-file. The option distancies instead of tracenos controls if the tracenumbers or the distances are used for the synchronization. With this option activated the distances defined in column 2 are used together with the distance coordinates of the file header in order to update the traceheader coordinates.

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

With the type **RAMAC-GPS** activated it is possible to read the GPS coordinates using the format of the RAMAC GPS device. The file should have the extension COR or CORC (cartesian coordinates). Each line of the GPS-ASCII file contains the following informations (following optional additional informations are ignored): *tracenumber,date,time,Northing,West,altitude,PDOP*

The separator between the different values may be either comma, blank or tab. In addition the orientation may follow the coordinate value seaparated by a comma, blank, tab or nothing (e.g.53.5891830N or 53.5891830,N).

The latitude coordinates are always set to negative values for Northing equal 'S' or Northing equal 'N' and negative input values. The longitude coordinates are always set to negative values for West equal 'W' or West equal 'E' and negative input values.

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

With the type **WSKTRANS-GPS** activated it is possible to read the GPS coordinates using the format of the WSKTRANS software. The file should have the extension ut or utm or tsp. The file consists of optional header lines and then each line of the GPS-ASCII file contains the following informations (following optional additional informations are ignored):

tracenumber Northing West altitude time(optional)

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

With the type **CSV-GPS** activated it is possible to read the GPS coordinates using the format of the CSV-GPS software. The file should have the extension csv. The file consists of one header line and then each line of the GPS-ASCII file contains the following informations:

latitude, longitude, altitude, tracenumber

Latitude and longitude must be given in the in the format ddmm.mmmm except the option **NMEA-string no degree** has been activated (then the coordinates will be imported as they are).

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

With the type **IDS-GPS** activated it is possible to read the GPS coordinates using the format of the IDS-GPS software. A GPS ASCII-file with the extension gps containing the GPS-coordinates must exist. The corresponding tracenumbers are taken from the comment markers with the comment "gps". Normally the original IDS-file contains these comment markers and they are taken over into the Reflexw file during import. Older IDS-systems generate an additional ASCII-file with the extension gpt containing the tracenumbers. If this file exists (same filename like the gps-file except the extension) the tracenumbers are taken from this ASCII-file. If the IDS-data have been acquired using the Stream software the secondary channels do not contain any informations about the gps-markers. If the conversion sequence has been set to multichannel the gps-markers will be taken over from the first channel data in this case.

There might be a header within the gps-file but the first line containing the first GPS-coordinates corresponds to the first tracenumber stored within comment marker or within the first line of the gpt-file. The next line corresponds to the second tracenumber and so on.

One line of the gps-file contains the following informations:

*\$GPGGA,123015.00,5141.607563,N,00439.680658,E,2,04,2.3,2.74,M,47.21,M,2.8,0000*40*

whereby the following fields will be interpreted:

field 3: 5141.607563 - geographic latitude in *ddmm.mmmmmmm* format (51 degrees and 41.607563 minutes)

field 5: 00439.680658 - geographic longitude in *dddmm.mmmmmmm* format (4 degrees and 39.680658 minutes)

field 10: 2.74 - antenna height above MSL (mean sea level) reference (2.74 m)

The latitude and the longitude are stored in degrees within the x- and y-receiver traceheader positions. The latitude coordinates are always set to negative values for Northing equal 'S' or Northing equal 'N' and negative input values. The longitude coordinates are always set to negative values for West equal 'W' or West equal 'E' and negative input values.

REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined.

With the option **consider blank lines** activated blank lines within the gps-file will be considered to be associated with a GPS-mark within the radar datafile.

Example:

```
$GP GGA,212821.30,4621.65127093,N,00611.07161442,E,4,19,0.6,459.011,M,48.164,M,1.3,0439*42
$GP GGA,212821.40,4621.65103360,N,00611.07122060,E,4,19,0.6,459.016,M,48.164,M,1.4,0439*4F
$GP GGA,212821.50,4621.65079547,N,00611.07082498,E,4,19,0.6,459.024,M,48.164,M,1.5,0439*49
```

```
$GP GGA,212821.70,4621.65031574,N,00611.07002490,E,4,19,0.6,459.034,M,48.164,M,0.7,0439*45
$GP GGA,212821.80,4621.65007531,N,00611.06961940,E,4,19,0.6,459.040,M,48.164,M,0.8,0439*4F
```

```
$GP GGA,212822.10,4621.64934322,N,00611.06839377,E,4,19,0.6,459.063,M,48.164,M,1.1,0439*42
```

```
$GP GGA,212822.30,4621.64885284,N,00611.06756595,E,4,19,0.6,459.081,M,48.164,M,1.3,0439*44
$GP GGA,212822.40,4621.64860710,N,00611.06715036,E,4,19,0.6,459.093,M,48.164,M,1.4,0439*4F
```

The program automatically controls if the number of GPS marks and the number of lines within the GPS-file correspond. If not a warning message appears and if blank lines within the gps-file exist it might be useful to activate the option **consider blank lines**

With the type **IDS-times** activated the gps-times for the individual traces are read from an ASCII-file created by IDS in the following form (here for a 4 channel system):

```
Chn;PS;Packet;Encoder;ClockCounter;Status;YYYYMMDD;TimeStampHHMMSS
1;0;1;65516;390732;1;20130829;070627.907320
2;0;2;65516;390870;1;20130829;070627.908700
3;0;3;65516;391008;1;20130829;070627.910080
4;0;4;65516;391147;1;20130829;070627.911470
1;1;5;65496;403564;1;20130829;070628.035640
2;1;6;65496;403703;1;20130829;070628.037030
3;1;7;65496;403841;1;20130829;070628.038410
4;1;8;65496;403979;1;20130829;070628.039790
```

Column 1 defines the channel number, 2 defines the tracenummer and column 8 the gps-time.

The program performs an automatic interpolation or extrapolation if no data are present for some traces.

These gps-times may be used for importing gps-data using the type **gps-times**.

With type **IDS-GEOX** activated the relative coordinates are read from the geox-files. If present the file UserMarkers.dat will be used for the xy reference coordinate.

```
<01.01.000>
<XYZLatLonAltTL>
764
```

```
1,-.331284451472892,.39801794668614,1,0,0,.517849083876708
2,-.372188774882243,.396415705932969,1,0,0,4.09356916273305E-02
```

```

3,-.415734322142434,.395758968497921,1,0,0,4.35504993111233E-02
4,-.46195840176718,.395337384737874,1,0,0,4.62260020986197E-02
5,-.510303558617004,.394877267426637,1,0,0,4.83473463467663E-02
6,-.558162396879843,.393745335548008,1,0,0,4.78722223178161E-02
7,-.604604194717226,.391815503848043,1,0,0,.046481876433278

```

REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined. Also available within the import and the edit several fileheader menu.

With type **IDS-GEAC** activated the relative or utm coordinates are read from the gec-files. In case of multichannel data there exists only one gec-file for the whole swath containing either the local or the utm coordinates for the whole swath.

Example:

```

<VERSION>
07.00.000
<OFFSET_GPS_ARRAY_XYZ>
0.13,0.42,0
<UTM_ZONE>
32,1
<REFERENCE_XYZ> converted longitude, converted latitude
512941.685563995,5290630.10008273,433.657
<REFERENCE_LLA>
0,0,0
<XCELL>
0.0416389
<VALID_MARKERS_SWEEP>
69

```

```

1,0,0,512941.685563995,5290630.10008273,433.657,32,1
8,0,1.59999999999854,512941.370795237,5290630.019358,433.667,32,1

```

REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined.

The update type is also available within the import and the edit several fileheader menu.

With the type **UTSI-GPS** activated it is possible to read the GPS coordinates using the format of the UTSI-GPS software. A GPS ASCII-file with the extension gps containing the GPS-coordinates must exist. The corresponding tracenumbers are either taken from this gps-file or from an additional ASCII-file with the extension gpt containing only the tracenumbers. If this file exists (same filename like the gps-file except the extension) the tracenumbers are taken from this ASCII-file. If it does not exist the gps-file must contain the tracenumbers in the following form:

```

tracenumber $GPGGA,123015.00,5141.607563,N,00439.680658,E,2,04,2.3,2.74,M,47.21,M,2.8,0000*40

```

If the additional gpt file exists the gps file does not contain the tracenumber:

```

$GPGGA,123015.00,5141.607563,N,00439.680658,E,2,04,2.3,2.74,M,47.21,M,2.8,0000*40

```

There might be a header within the gps-file but the first line containing the first GPS-coordinates corresponds to the given tracenumber of the first tracenumber

stored within the first line of the gpt-file. The next line corresponds to the second tracenummer and so on.

The following fields of the NMEA gps code will be interpreted:

field 3: 5141.607563 - geographic latitude in *ddmm.mmmmmmm* format (51 degrees and 41.607563 minutes)

field 5: 00439.680658 - geographic longitude in *dddmm.mmmmmmm* format (4 degrees and 39.680658 minutes)

field 10: 2.74 - antenna height above MSL (mean sea level) reference (2.74 m)

The latitude and the longitude are stored in degrees within the x- and y-receiver traceheader positions. The latitude coordinates are always set to negative values for Northing equal 'S' or Northing equal 'N' and negative input values. The longitude coordinates are always set to negative values for West equal 'W' or West equal 'E' and negative input values.

REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined.

With the type **PulseEkko-GPS** activated it is possible to read the GPS coordinates using the format of the PulseEkko-GPS software. Two different formats with the extensions gps and gp2 are supported.

Gps-format: Each block of the file starts with a line containing the trace number. A number of lines follow containing the gps-coordinates. By default the coordinates are taken from the line starting with \$GPGGA. Following an example of one GPS-block is shown:

Trace #1 at position 0.000000

\$GPVTG,266.2,T,,.007.12,N,013.19,K,A*4E

\$GPGSA,M,3,03,19,15,16,18,22,,,,,2.8,1.7,2.2*33

\$GPRMC,093704,V,4625.938530,N,00950.727816,E,007.12,266.2,240404,0.7,W,N*2D

\$GPGGA,093705.00,4625.938360,N,00950.724940,E,1,06,1.7,2635.92,M,48.00,M,,*53

gp2-format: a header of 6 lines follows a line containing the tracenummer and the GPS-block separated by two “-characters:

;GPS@@@

;Ver=1.1.0

;DIP=2009-00152-00

;Date=2015-Aug-26

traces,odo_tick,pos(m),time_elapsed(s),GPS

1,0,0,0,"\$GPGGA,112349,2916.7149155,S,3126.59769646,E,1,1,0.1,4.88,M,26.68,M,0,0"

300,0,0,0,"\$GPGGA,112610,2916.7219823,S,3126.61275112,E,1,1,0.1,3.17,M,26.68,M,0,0"

For both formats within the \$GPGGA line the following fields will be interpreted:

field 3: 4625.938360 - geographic latitude in *ddmm.mmmmmmm* format (46 degrees and 25.938360 minutes)

field 5: 000950.727816 - geographic longitude in *dddmm.mmmmmmm* format (9 degrees and 50.727816 minutes)

field 10: 2635.9 - antenna height above MSL (mean sea level) reference (2635.92 m)

The latitude and the longitude are stored in degrees within the x- and y-receiver traceheader positions. The latitude coordinates are always set to negative values for Northing equal 'S' or Northing equal 'N' and negative input values. The longitude coordinates are always set to negative values for West equal 'W' or West equal 'E' and negative input values.

REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined.

With the type **GSSI-GPS** activated it is possible to read the GPS coordinates using the format of the GSSI-GPS software. One GPS file with the extension dzg must exist. Each block of the file consists of two lines, one with the tracenummer code starting with \$GSSIS followed by the line containing the GPS coordinates starting with \$GPGGA. Following an example of one GPS-block is shown:

\$GSSIS,127.0

\$GPGGA,183836.20,5254.4730,N,00114.8307,W,1,15,0.8,42.67,M,49.10,M,*,42

Within the \$GPGGA line the following fields will be interpreted:

field 3: 5254.4730 - geographic latitude in *ddmm.mmmmmmm* format (52 degrees and 54.4730 minutes)

field 5: 00114.8307 - geographic longitude in *dddmm.mmmmmmm* format (1 degrees and 14.8307 minutes)

field 10: 42.67 - antenna height above MSL (mean sea level) reference (42.67 m)
The latitude and the longitude are stored in degrees within the x- and y-receiver traceheader positions. The latitude coordinates are always set to negative values for Northing equal 'S' or Northing equal 'N' and negative input values. The longitude coordinates are always set to negative values for West equal 'W' or West equal 'E' and negative input values.

REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined.

The option **latitude** (x-coord./y-coord.) specifies where the original latitude (or 1. coordinate in general) shall be stored - either on the x- or the y-position within the Reflexw traceheader. The option is available for all GPS-types. Setting the option to y-coord. might be useful if you are using for example the option view/profile line in order to have the same map orientation as in normal geographic maps. The actual setting is saved when leaving the program.

The option **trace delay** available for 'RAMAC-GPS', 'WSKTRANS-GPS' and 'CSV-GPS' format allows to specify a delay number for the trace numbers given within the gps-file. Enter a number greater than 0 if the gps is delayed. This number will be subtracted from each given tracenummer within the gps-file.

The option **NMEA-string no degree** available for 'IDS-GPS', 'PulseEkko-GPS', 'UTSI-GPS', 'CSV-GPS', 'GEODE-GPS', 'GSSI-GPS', 'Marker-GPS', 'GPS-times' or 'GPS-times spatial interpol.' allows if activated that the \$GP Strings do not contain degree values but the string will be interpreted as it is.

Special NMEA String: With \$GNLLQ activated within the global settings menu a special GNLLQ string will be analysed for gps data with NMEA string. This string normally contains Gauss Krueger coordinates. In this case the option NMEA-string no degree option should be activated in addition.

With the type **Marker-GPS** activated it is possible to synchronize the GPS with markers stored with the datafile.

The program assumes that each marker corresponds to one line within the gps-datafile and the that first gps-coordinate line fits the first marker.

With the option **M-Master format** activated the format of the M-Master software.

One GPS file with the extension txt must exist. Two header lines must be present. Each line of the GPS-ASCII file contains the following informations (following optional additional informations are ignored):

markernumber,x-coordinate,y-coordinate,latitude,longitude,distance

The separator between the different values may be either comma, blank or tab.

The latitude and longitude coordinates will be ignored. The markers must be present within the datafile.

Example:

M.Nr.	X	Y	Latitude	Longitude	Length
1	7439138	1674843	66.99803	19.81500	0
2	7439128	1674847	66.99794	19.81516	10
3	7439117	1674852	66.99784	19.81526	22
4	7439105	1674858	66.99773	19.81539	35
5	7439090	1674864	66.99759	19.81551	51
6	7439074	1674872	66.99744	19.81565	68
7	7439056	1674879	66.99728	19.81580	87
8	7439039	1674888	66.99712	19.81597	106
9	7439020	1674895	66.99695	19.81611	126
10	7439002	1674904	66.99678	19.81626	145

File created 4/18/07 6:04:35 PM

REFLEXW stores the X- and y-coordinates and the distance into the traceheaders at the corresponding marker position using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing markers) or where the GPS-coordinates are not defined.

With the option **NWI-GPS** activated ASCII-data using the format

tracenummer dummy date time longitude latitude

17 (0) 28.04.2009 15:48:26 <644.482580><5357.605670>(2)

will be used. The latitude and the longitude are stored in degress within the x- and y-receiver traceheader positions.

REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined.

With the option **Geode-GPS** activated the GPS coordinates using the format of the Geode software are imported. One GPS file with the extension log must exist. Each block of the file starts with a line containing the trace number. A header with 2 or more lines may exist followed by a number of GPS blocks consisting of one line containing the GPS coordinates and one line containing the FFID

number (field record number) which is used for the assignment to the traceheader field record number of the original data. Following an example of one GPS-block is shown:

\$GPGGA,162241.00,4625.938360,N,000950.727816,E,2,10,1.0,2635.9,M,45.6,M,5.0,0120*46
FFID 3783 (Stack 1, Shot Loc: 0 Meters) 16:22:41.00 09/28/2010 160 KBytes SAVED IN 3782.SGY

Within the \$GPGGA line the following fields will be interpreted:

field 3: 4625.938360 - geographic latitude in *ddmm.mmmmmmm* format (46 degrees and 25.938360 minutes)

field 5: 000950.727816 - geographic longitude in *dddmm.mmmmmmm* format (9 degrees and 50.727816 minutes)

field 10: 2635.9 - antenna height above MSL (mean sea level) reference (2635.92 m)

The coordinates will be assigned to those traces which field record number within the traceheader is identical to 3783.

The latitude and the longitude are stored in degree within the x- and y-receiver traceheader positions.

REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined.

The option is also available within the import menu.

With the option **DB-GPS** activated the GPS coordinates using the format of the DB Engineering software are imported:

TimeTick-Korrekturwerte:	636297668240000000	10000			
km	Scan	Latitude	Longitude	Höhe	TimeTick
85,805	1	5211,6893871	842,4050947	109,361	636297668255000000
85,815	204	5211,6865581	842,3970449	109,354	636297668321000000
85,825	402	5211,6839247	842,3895242	109,353	636297668347000000

The latitude and the longitude are stored in degree within the x- and y-receiver traceheader positions.

REFLEXW stores the latitude, the longitude and the altitude into the traceheaders using a 32 or 64 bit floating format depending on the output format during the import of the data. If there are consecutive identical coordinates in the gps-file these coordinates are automatically be replaced with an interpolation. An interpolation or extrapolation is also done where no coordinates are defined (missing tracenumbers) or where the GPS-coordinates are not defined.

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

The options **GPS-times** and **GPS-times spatial interpolation** allow to update the traceheader coordinates based on the GPS acquisition times (in secs after midnight) stored within the file traceheader and an ASCII-file containing the GPS-coordinates and the GPS-times.

GPS-times should be used if the data acquisition has been done on a time-base. In this case first an automatic interpolation of the GPS-times is done at those positions where no acquisition times had been defined.

GPS-times spatial interpolation should be used if the data acquisition has been done on a distance base, e.g. using a wheel. No interpolation of the GPS-times will be done and only at those where the acquisition times have been defined the GPS-coordinates will be placed. Afterwards a spatial interpolation of the coordinates will be done at those positions where no acquisition times had been defined.

Five different ASCII-formats for the GPS-data are available:

NMEA-Format, ASCII1-Format, ASCII-MapSource, GPX and Proceq.

The NMEA-format follows the NMEA GPS standard. If Proceq is activated the two NMEA sentences \$GNGGA and \$GPGGA are supported. Otherwise two types of messages are supported:

\$GPRMC,093704,V,4625.938530,N,00950.727816,E,007.12,266.2,240404,0.7,W,N*2D

\$GPGGA,093705.00,4625.938360,N,00950.724940,E,1.06,1.7,2635.92,M,48.00,M,,*53

In case of a \$GPGGA sentence the data for a GPS-quality of 0 will be ignored.

The format of the ASCII1-file is as following:

Easting	Northing	MSL	GPS Date	GPS Time
638434601	7215215.236	143.227	19.05.02	07:32:12
638434.573	7215215.253	143.191	190502	0.3140509259

There is no header line - the ASCII-file must begin with the first data values. The GPS-date must be present but it will not be used for the matching. Only the GPS-times are used.

Two different formats for the ASCII-MapSource for UTM-data and Gauss Krüger data are supported:

UTM:

Trackpoint 32 U 471164 5443567 24.11.2009 17:15:53 103 m 5.03 m 00:00:05 3.6 kph 162° true

Trackpoint 32 U 471159 5443560 24.11.2009 17:15:58 105 m 8.56 m 00:00:05 6.2 kph 213° true

Gauss Krüger:

Trackpoint 3 471229 5445306 24.11.2009 17:15:53 103 m 5.03 m 00:00:05 3.6 kph 162° true

Trackpoint 3 471224 5445299 24.11.2009 17:15:58 105 m 8.56 m 00:00:05 6.2 kph 213° true

The seconds within the time may also have decimal places.

The format GPX analyses the trkpt string of the following form:

<trkpt lat="47.1052278858" lon="12.7156866621"> <ele>2932.64</ele>

<time>2016-09-14T06:58:33Z</time></trkpt>

The program performs an automatic interpolation or extrapolation if the times do not exactly match (normally the case). Only those data which lie inbetween the timerange defined by the first and last trace of the profile are taken into account. This overcomes the problem that GPS-data would be taken into account at times when the acquisition unit did not move (e.g. at the start or the end of the data acquisition).

With the suboption use lineparts activated only the times stored for the first ensemble will be used for the synchronization with the external GPS-file. The found coordinates will be transferred to the corresponding traceheaders of the other ensembles. Activate this option if the file is a 3D-datafile consisting of parallel 2D-files stored sequentially. As a result all 2D-ensembles will have the same coordinates - this means that the offset between the 2D-lines will not be taken into account.

With the suboption max.dev.time 1s activated acquisition times and GPS times are not synchronized if the deviation between them is more than 1 sec. In this

case these coordinates will remain zero and a succeeding linear interpolation at those positions will be done. The option is also available within the dataimport and the edit several FileHeaders menu.

The program also supports the use if the GPS acquisition times (in secs after midnight) are not stored within the file traceheader. For that purpose the following assumptions must hold true:

- the data have been acquired using an equidistant timebase (traces per second are identical within the complete profile)
- the start and end time of the profile is identical to the start and end time of the GPS data

If necessary the start and end time of the GPS data must be adapted manually.

Two different types of interpolation are supported:

- with the option pure linear interpolation active the interpolation is linear for all data
- with the option pure linear interpolation deactivated the interpolation is linear if the mean acquisition velocities between two successive GPS time marks do not vary more than 10 %. Otherwise the program performs an interpolation based on a constant acceleration. The option check accelerations smoothes the calculated accelerations. Activate this option if the resulting interpolations show too strong undulations.

The option **calculate distancies** allows to update the traceheader distances based on the x- and y-receiver traceheader coordinates and optionally (option use z-coordinates activated) based on the z-receivertraceheader coordinates in addition. The start distance is taken from the fileheader but can be changed manually (option start distance). The distances will be summed up from trace to trace.

With the option straight line from start point activated the distances will be calculated as a straight line from the start point. This might be useful if the gps coordinates vary irregularly from trace to trace and if the profile was acquired along a more or less straight line.

For the datatypes single shot and borehole.... the distances are taken from: receiver x-coordinate for single shot
receiver y- or z-coordinate for borehole data depending on the setting of the option use z-coordinate.

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

Using the option **multiply by constant value** allows to multiply the xy- shot and receiver coordinates by one constant value.

When using the option **exchange rec. -> distance** the distance traceheadercoordinate is set to the x-receiver coordinate for each trace. Afterwards the x- and y-receiver coordinates are set to the corresponding shot coordinaes.

The option **CopyCMPCoordToShotRecCoord** allows to copy the CMP-coordinates to the source and receiver coordinates.

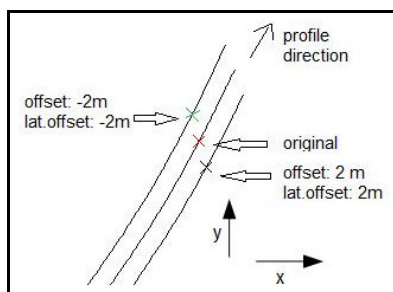
The option **get ensemble/scandir.from coord.** extracts both the scandirection and the ensemble number of a constant offset or 3D-constant offs.file from the given traceheader receiver coordinates. The scan direction will be set to 1 if

successive y-coordinates do not change and set to 2 if successive x-coordinates do not change. The ensemble number will be updated if the profile constant coordinate changes.

The option **project on x-coord.** calculates the distances along the line based on the receiver coordinates (see also option calculate distances) and projects these distances onto the x-axis: the x-receiver coordinates are set to these distances and the y-coordinates are set to 0. The distance of the shot is also automatically determined and the shot x-coordinate is set to this distance. This is not strictly correct if the shot is not situated within the receiver line. The start distance is queried after having activated the option.

The option does not work correctly if the profile consists of several shots with different receivers.

The option might be useful if GPS-coordinates are present within the traceheaders. These coordinates cannot be used for most of the refraction interpretation tools which need the coordinates only along the x-axis.



The option **correct for offset in profile direction** allows to correct the xy-GPS coordinates for an offset in profile direction (option **offset**) and perpendicular to it (option **lateral offset**). The option might be useful if there exists any offset between the geophones/shots and the GPS-unit, e.g. a data acquisition on a ship with a boomer and hydrophones.

The **offset** in profile direction will be subtracted

from the distances in profile direction (this means a negative value must be entered if the GPS unit is located behind the receivers in profile direction and a positive value must be entered if the GPS unit is located before). A positive **lateral offset** results in a shift to the right of the profile direction, a negative lateral offset results in a shift to the left.

Two different cases for the automatic definition of the profile direction are considered:

1. Special case if the GPS and the source/receiver unit are exactly located on the same track. For this the following must hold true:

- the distances within the traceheader are specified (see also "update distances") or activating the option **use fileheader dist.** which uses the fileheader distances instead of the traceheader distances for the pure correction in profile direction.

- the lateral offset is set to 0

- "smooth dir." is set to back

Now the correction is done based on the trace header distances (see also update option calculate distances). This means that the corrected values follow exactly the track of the GPS-unit. If an extrapolation is necessary at the end or at the start of the profile (depending on the correct value) the program calculates the actual direction of the profile from the difference of the actual GPS-coordinate and the previous one. In this case also the z-coordinates are updated.

2. GPS and source/receiver do not exactly follow the same track. In this case the z-coordinates will not be updated. The option **smooth dir.** together with the entered number of **smooth traces**. control the automatic definition of the profile direction from the GPS coordinates for the subsequent correction. The options **back, middle and forward** allow to simulate different coupling mechanism of GPS and source/receiver devices.

Back might be useful for a flexible connection with the GPS before the source/receive device, middle for a rigid one and forward for a flexible connection with the GPS behind the source/receiver device.

The option **offsets from file** allows to read the correct offsets (in profile direction - offset - and perpendicular to it - lat.offset) from an ASCII-file named CorrectOffset.txt which must have been stored under the subdirectory ASCII under the actual project directory. Each line of the file must contain the following parameters:

“channel number” “offset” “lateral offset”

Example of 3 channels with a symmetrical array:

1 0.5 0.5

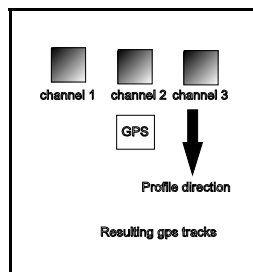
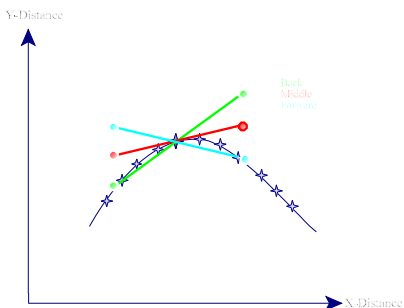
2 0.5 0.0

3 0.5 -0.5

For a **3D-const.offset** file the channelnumber corresponds to the individual ensemble numbers of the 3D-file. Thereby the correct offset file must contain the complete geometry of the antenna array. This holds true for example for a 3D-file which has been acquired using an antenna array like 3D-radar.

The option is also available within the “Edit several fileheaders” menu.

It is recommended to check the correction, e.g. using the option view/profile line. To be considered: the z-coordinates will also be updated if no interpolation is necessary, this means the correction is only done in profile direction (lateral offset 0 and smooth direction set to back). Precondition is that the traceheader distances have been defined before (option calculate distances).



The option **correct latency** allows to correct a possible gps latency. Precondition is that the gps times have been stored within the Reflexw traceheaders together with the gps coordinates. GPS always has the problem of latency when determining position, which is why it cannot output an exact real-time position. Different GPS can have different latency. If this time delay has not been considered during the gpr/gps synchronization it may be corrected manually using this option. The determination of the latency is not easy and there are no general rules and the individual conditions must be considered. The entered latency in s will be subtracted from the times stored within the Reflexw traceheaders (normally coming from the synchronized original gps file) and this new value is the base for searching the corresponding gps coordinates which replace the old ones. Normally the latency will be given in positive values and thereby at the beginning of the gpr line no gps coordinates are present. Here an automatic extrapolation will be done.

The option **topography (x,z values)** reads the topography (shot and receiver z-values) from an ASCII-file. Each line of the ASCII-file contains the distance and the depth or elevation. The option allows the independent input of the topography for the shot and the receiver coordinates (options update shot z-pos. and update receiver z-pos.). With activated option use x-traceheadercoord. the x-receiver traceheader coordinate (rec-x) is used and with deactivated option the distance within the traceheader is used. A linear interpolation or extrapolation is done where no values are defined within the ASCII-file.

With the option apply x-z topography activated the geometries of the shots and the receivers are recalculated based on the topographic xz-values. 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 option is identical to the option apply x-z topography within the traveltimes analysis module with the difference that the z-values must be given as depths. It is assumed that the x-values of the data do not represent the correct x-coordinates but are determined directly on the topographic interface. The program automatically determines the positions of all receivers on the given topography and calculates the x- and z-projections of these positions. The x-coordinate within the ASCII-file represents the true x-coordinate within the xz-coordinate system.

The following assumptions must hold true:

1. The topography file must contain the values for x=0 (fixed point) because this value serves at the starting point for determining the corrections. All corrections are relative to the value at x=0.
2. The profiledirection is pure x, this means that the y-coordinates of all shots and receivers are equal.
3. The original x-values are taken directly on the topographic interface.

The application of this option is useful if you want to introduce a topography for the wavefront-inversion or for the raytracing (chap. 4.7).

The option **second profile marker positions** allows to copy the traceheader coordinates and times from a secondary profile at the given marker positions of both profiles. After having activated the option the filename of the secondary profile will be queried. Precondition is that the marker positions are identical. An application might be the synchronization of a newer acquisition system using a gps unit and an older system which cannot be synchronized with a gps unit.

The option **UTM-conversion** allows to convert GPS degree traceheader coordinates to local cartesian coordinates using the so called Universal-Transverse-Mercator(UTM)-conversion (WGS84). If the distancedimension = 'foot' the utm-conversion gives foot-values otherwise meter values.

You must specify where the latitude is stored within the traceheader (latitude in degrees either in x- (shot x, rec x and CMP x-pos) or y-coord). The longitude is automatically vice versa in y- or x-coord.

The eastern longitude values must be positive, the western longitudes values must be negative (see zones below).

The northern latitude values must be positive, the southern latitude values must be negative (see zones below).

You can specify if the original data are in degrees or in arcsecs. If arcsecs is chosen the original data are automatically divided by 3600 in order to transform them into degrees.

The conversion can be done automatically (default) or by using a distinct zone. If the automatic conversion is used, the UTM-zone for the whole profile is determined using the GPS traceheader coordinates of the first trace. So the

coordinate origin is fixed for the whole profile not regarding, that the profile may lie in different UTM-zones. The irregular utm-zones will also be automatically taken into account. These are the zone 32V (South Norway) and 31X, 33X, 35X, 37X (Svalbard).

If a distinct zone shall be used for the conversion, it can be chosen from a list if the option 'distinct zone' is chosen. This zone is also fixed for the whole profile. There exist 120 different zones which cover nearly the whole globe: 60 zones cover the northern part of the world ($0 \text{ deg} \leq \text{lat} \leq 84 \text{ deg}$), the rest the southern part ($-80 \text{ deg} \leq \text{lat} < 0 \text{ deg}$).

The zones of every half sphere are determined as following:

Zone 1: $-180 \text{ deg} \leq \text{lon} \leq -174 \text{ deg}$ (if $\text{lat} \geq 0$ this is zone utm1n, otherwise utm1s)

Zone 2: $-174 \text{ deg} < \text{lon} \leq -168 \text{ deg}$

..

Zone 60: $174 \text{ deg} < \text{lon} \leq 180 \text{ deg}$ (if $\text{lat} < 0$ this is zone utm60s, otherwise utm60n)

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

The following table contains a complete listing of all UTM zones and their association to the countries of the world, so that the appropriate zone can be chosen for the manual conversion:

UTM zones and their association to the countries of the world:

UTM zone 1N	-180deg to -174deg Northern hemisphere. Russia.
UTM zone 1S	-180deg to -174deg Southern hemisphere.
UTM zone 2N	-174deg to -168deg Northern hemisphere. Russia. United States (USA) - Alaska (AK).
UTM zone 2S	-174deg to -168deg Southern hemisphere.
UTM zone 3N	-168deg to -162deg Northern hemisphere. United States (USA) - Alaska (AK).
UTM zone 3S	-168deg to -162deg Southern hemisphere.
UTM zone 4N	-162deg to -156deg Northern hemisphere. United States (USA) - Alaska (AK).
UTM zone 4S	-162deg to -156deg Southern hemisphere.
UTM zone 5N	-156deg to -150deg Northern hemisphere. United States (USA) - Alaska (AK).
UTM zone 5S	-156deg to -150deg Southern hemisphere.
UTM zone 6N	-150deg to -144deg Northern hemisphere. United States (USA) - Alaska (AK).
UTM zone 6S	-150deg to -144deg Southern hemisphere.
UTM zone 7N	-144deg to -138deg Northern hemisphere. Canada - British Columbia (BC); Yukon. United States (USA) - Alaska (AK).
UTM zone 7S	-144deg to -138deg Southern hemisphere.
UTM zone 8N	-138deg to -132deg Northern hemisphere. Canada - British Columbia (BC); North West Territories; Yukon. United States (USA) - Alaska (AK).
UTM zone 8S	-138deg to -132deg Southern hemisphere.
UTM zone 9N	-132deg to -126deg Northern hemisphere. Canada - British Columbia (BC); North West Territories; Yukon. United States (USA) - Alaska (AK).
UTM zone 9S	-132deg to -126deg Southern hemisphere.

UTM zone 10N	-130deg to -120deg Northern hemisphere. Canada - British Columbia (BC); North West Territories; Yukon. United States (USA) - Alaska (AK).
UTM zone 10S	-130deg to -120deg Southern hemisphere.
UTM zone 11N	-120deg to -114deg Northern hemisphere. Canada - Alberta; British Columbia (BC); North West Territories. Mexico. United States (USA).
UTM zone 11S	-120deg to -114deg Southern hemisphere.
UTM zone 12N	-114deg to -108deg Northern hemisphere. Canada - Alberta; North West Territories; Saskatchewan. Mexico. United States (USA).
UTM zone 12S	-114deg to -108deg Southern hemisphere.
UTM zone 13N	-108deg to -102deg Northern hemisphere. Canada - North West Territories; Saskatchewan. Mexico. United States (USA).
UTM zone 13S	-108deg to -102deg Southern hemisphere.
UTM zone 14N	-102deg to -96deg Northern hemisphere. Canada - Manitoba; North West Territories; Saskatchewan. Mexico. United States (USA).
UTM zone 14S	-102deg to -96deg Southern hemisphere.
UTM zone 15N	-96deg to -90deg Northern hemisphere. Canada - Manitoba; North West Territories; Ontario. Guatemala. Mexico. United States (USA) - Gulf of Mexico (GoM).
UTM zone 15S	-96deg to -90deg Southern hemisphere.
UTM zone 16N	-90deg to -84deg Northern hemisphere. Belize. Canada - Manitoba; North West Territories; Ontario. Costa Rica. Cuba. El Salvador. Guatemala. Honduras. Mexico. Nicaragua. Puerto Rico. United States (USA).
UTM zone 16S	-90deg to -84deg Southern hemisphere.
UTM zone 17N	-84deg to -78deg Northern hemisphere. Bahamas. Ecuador - north of equator. Canada - North West Territories; Ontario; Quebec. Cayman Islands. Colombia. Costa Rica. Jamaica.
UTM zone 17S	-84deg to -78deg Southern hemisphere. Ecuador. Peru.
UTM zone 18N	-78deg to -72deg Northern hemisphere. Bahamas. Canada - North West Territories; Ontario; Quebec. Colombia. Cuba. Ecuador. Greenland. Haiti. Jamaica. Panama. Turks and Caicos Islands. United States (USA). Venezuela.
UTM zone 18S	-78deg to -72deg Southern hemisphere. Argentina. Brazil. Chile. Colombia. Ecuador. Peru.
UTM zone 19N	-72deg to -66deg Northern hemisphere. Aruba. Bahamas. Brazil. Canada - New Brunswick (NB); Newfoundland; North West Territories; Nova Scotia (NS); Quebec. Colombia. Dominican Republic. Greenland. Netherlands Antilles. Turks and Caicos Islands. United States. Venezuela.
UTM zone 19S	-72deg to -66deg Southern hemisphere. Argentina. Bolivia. Brazil. Chile. Colombia. Peru.
UTM zone 20N	-66deg to -60deg Northern hemisphere. Anguilla. Antigua & Barbuda. Bermuda. Brazil. Canada - New Brunswick (NB); Newfoundland; North west Territories; Nova Scotia (NS); Prince Edward Island; Quebec. Dominica. Greenland. Grenada. Guadeloupe. Guyana. Martinique. Montserrat. Saint Kitts and Nevis. Saint Lucia. Saint Vincent and the Grenadines. Trinidad and Tobago. Venezuela. Virgin islands.
UTM zone 20S	-66deg to -60deg Southern hemisphere. Argentina. Bolivia. Brazil. Falkland Islands (Malvinas). Paraguay.
UTM zone 21N	-60deg to -54deg Northern hemisphere. Barbados. Brazil. Canada - Newfoundland; Quebec. French Guiana. Greenland. Guyana. St. Pierre and Miquelon. Suriname.
UTM zone 21S	60deg to 54deg Southern hemisphere. Argentina. Bolivia. Brazil.

	Falkland Islands (Malvinas). Paraguay. Uruguay.
UTM zone 22N	-54deg to -48deg Northern hemisphere. Brazil. Canada - Newfoundland. French Guiana. Greenland.
UTM zone 22S	-54deg to -48deg Southern hemisphere. Brazil. Uruguay.
UTM zone 23N	-48deg to -42deg Northern hemisphere. Greenland.
UTM zone 23S	-48deg to -42deg Southern hemisphere. Brazil.
UTM zone 24N	-42deg to -36deg Northern hemisphere. Greenland.
UTM zone 24S	-42deg to -36deg Southern hemisphere. Brazil. South Georgia and the South Sandwich Islands.
UTM zone 25N	-36deg to -30deg Northern hemisphere. Greenland.
UTM zone 25S	-36deg to -30deg Southern hemisphere. Brazil.
UTM zone 26N	-30deg to -24deg Northern hemisphere. Greenland. Iceland.
UTM zone 26S	-30deg to -24deg Southern hemisphere.
UTM zone 27N	-24deg to -18deg Northern hemisphere. Greenland. Iceland.
UTM zone 27S	-24deg to -18deg Southern hemisphere.
UTM zone 28N	-18deg to -12deg Northern hemisphere. Gambia. Greenland. Guinea. Guinea-Bissau. Iceland. Ireland - offshore Porcupine Basin. Mauritania. Morocco. Senegal. Sierra Leone. Western Sahara.
UTM zone 28S	-18deg to -12deg Southern hemisphere.
UTM zone 29N	-12deg to -6deg Northern hemisphere. Algeria. Cote D'Ivoire. Faroe Islands. Guinea. Ireland. Mali. Mauritania. Morocco. Portugal. Sierra Leone. Spain. United Kingdom (UK). Western Sahara.
UTM zone 29S	-12deg to -6deg Southern hemisphere.
UTM zone 30N	-6deg to 0deg Northern hemisphere. Algeria. Burkino Faso. Cote' Ivoire. Faroe Islands - offshore. France. Ghana. Gibraltar. Ireland - offshore Irish Sea. Mali. Mauritania. Morocco. Spain. United Kingdom (UK).
UTM zone 30S	-6deg to 0deg Southern hemisphere.
UTM zone 31N	0deg to 6deg Northern hemisphere. Algeria. Andorra. Benin. Burkino Faso. Denmark - North Sea. France. Germany - North Sea. Ghana. Luxembourg. Mali. Netherlands. Niger. Nigeria. Norway.
UTM zone 31S	0deg to 6deg Southern hemisphere.
UTM zone 32N	6deg to 12deg Northern hemisphere. Algeria. Austria. Cameroon. Denmark. Equatorial Guinea. France. Gabon. Germany. Italy. Libya. Liechtenstein. Monaco. Netherlands. Niger. Nigeria. Norway. Sao Tome and Principe. Svalbard and Jan Mayen Islands. Sweden. Switzerland. Tunisia. Vatican City State.
UTM zone 32S	6deg to 12deg Southern hemisphere. Angola. Congo. Gabon. Namibia.
UTM zone 33N	12deg to 18deg Northern hemisphere. Austria. Bosnia and Herzegowina. Cameroon. Central African Republic. Chad. Congo. Croatia. Czech Republic. Democratic Republic of the Congo (Zaire). Gabon. Germany. Hungary. Italy. Libya. Malta. Niger. Nigeria. Norway. Poland. San Marino. Slovakia. Slovenia. Svalbard and Jan Mayen Islands. Sweden.
UTM zone 33S	12deg to 18deg Southern hemisphere. Angola. Congo. Democratic Republic of the Congo (Zaire). Gabon. Namibia. South Africa.
UTM zone 34N	18deg to 24deg Northern hemisphere. Albania. Belarus. Bosnia and Herzegowina. Bulgaria. Central African Republic. Chad. Croatia. Democratic Republic of the Congo (Zaire). Estonia. Finland. FYR Macedonia. Greece. Hungary. Italy. Latvia. Libya. Lithuania. Norway. Poland. Romania. Russia. Slovakia. Sudan. Svalbard. Sweden. Ukraine. Yugoslavia.
UTM zone 34S	18deg to 24deg Southern hemisphere. Angola. Botswana.

	Democratic Republic of the Congo (Zaire). Namibia. South Africa. Zambia.
UTM zone 35N	24deg to 30deg Northern hemisphere. Belarus. Bulgaria. Central African Republic. Democratic Republic of the Congo (Zaire). Egypt. Estonia. Finland. Greece. Latvia. Lesotho. Libya. Lithuania. Moldova. Norway. Romania. Russia. Sudan. Svalbard. Turkey. Uganda. Ukraine.
UTM zone 35S	24deg to 30deg Southern hemisphere. Botswana. Burundi. Democratic Republic of the Congo (Zaire). Rwanda. South Africa. Tanzania. Uganda. Zambia. Zimbabwe.
UTM zone 36N	30deg to 36deg Northern hemisphere. Belarus. Cyprus. Egypt. Ethiopia. Finland. Israel. Jordan. Kenya. Lebanon. Norway. Russia. Saudi Arabia. Sudan. Syria. Turkey. Uganda. Ukraine.
UTM zone 36S	30deg to 36deg Southern hemisphere. Burundi. Kenya. Malawi. Mozambique. Rwanda. South Africa. Swaziland. Tanzania. Uganda. Zambia. Zimbabwe.
UTM zone 37N	36deg to 42deg Northern hemisphere. Djibouti. Egypt. Eritrea. Ethiopia. Georgia. Iraq. Jordan. Kenya. Lebanon. Russia. Saudi Arabia. Somalia. Sudan. Syria. Turkey. Ukraine.
UTM zone 37S	36deg to 42deg Southern hemisphere. Kenya. Mozambique. Tanzania.
UTM zone 38N	42deg to 48deg Northern hemisphere. Armenia. Azerbaijan. Djibouti. Eritrea. Ethiopia. Georgia. Iran. Iraq. Kazakhstan. Kuwait. Russia. Saudi Arabia. Somalia. Turkey. Yemen.
UTM zone 38S	42deg to 48deg Southern hemisphere. Madagascar.
UTM zone 39N	48deg to 54deg Northern hemisphere. Azerbaijan. Bahrain. Iran. Kazakhstan. Kuwait. Oman. Qatar. Russia. Saudi Arabia. Somalia. Turkmenistan. United Arab Emirates. Yemen.
UTM zone 39S	48deg to 54deg Southern hemisphere. Madagascar.
UTM zone 40N	54deg to 60deg Northern hemisphere. Iran. Kazakhstan. Oman. Russia. Saudi Arabia. Turkmenistan. United Arab Emirates. Uzbekistan.
UTM zone 40S	54deg to 60deg Southern hemisphere. Seychelles.
UTM zone 41N	60deg to 66deg Northern hemisphere. Afghanistan. Iran. Kazakhstan. Pakistan. Russia. Turkmenistan. Uzbekistan.
UTM zone 41S	60deg to 66deg Southern hemisphere.
UTM zone 42N	66deg to 72deg Northern hemisphere. Afghanistan. India. Kazakhstan. Kyrgyzstan. Pakistan. Russia. Tajikistan. Uzbekistan.
UTM zone 42S	66deg to 72deg Southern hemisphere.
UTM zone 43N	72deg to 78deg Northern hemisphere. China. India. Kazakhstan. Kyrgyzstan. Maldives. Pakistan. Russia. Tajikistan.
UTM zone 43S	72deg to 78deg Southern hemisphere.
UTM zone 44N	78deg to 84deg Northern hemisphere. China. India. Kazakhstan. Kyrgyzstan. Nepal. Russia. Sri Lanka.
UTM zone 44S	78deg to 84deg Southern hemisphere.
UTM zone 45N	84deg to 90deg Northern hemisphere. Bangladesh. Bhutan. China. India. Kazakhstan. Mongolia. Nepal. Russia.
UTM zone 45S	84deg to 90deg Southern hemisphere.
UTM zone 46N	90deg to 96deg Northern hemisphere. Bangladesh. Bhutan. China. Indonesia. Mongolia. Myanmar (Burma). Russia.
UTM zone 46S	90deg to 96deg Southern hemisphere.
UTM zone 47N	96deg to 102deg East; Northern hemisphere. China. Indonesia. Laos. Malaysia - West Malaysia. Mongolia. Myanmar (Burma). Russia. Thailand.

UTM zone 47S	96deg to 102deg East; Southern hemisphere. Indonesia.
UTM zone 48N	102deg to 108deg Northern hemisphere. Cambodia. China. Indonesia. Laos. Malaysia - West Malaysia. Mongolia. Russia. Singapore. Thailand. Vietnam.
UTM zone 48S	102deg to 108deg Southern hemisphere. Indonesia.
UTM zone 49N	108deg to 114deg Northern hemisphere. China. Hong Kong. Indonesia. Macau. Malaysia - East Malaysia - Sarawak. Mongolia. Russia. Vietnam.
UTM zone 49S	108deg to 114deg Southern hemisphere. Australia. Indonesia.
UTM zone 50N	114deg to 120deg Northern hemisphere. Brunei. China. Indonesia. Malaysia - East Malaysia - Sarawak. Mongolia. Philippines. Russia. Taiwan.
UTM zone 50S	114deg to 120deg Southern hemisphere. Australia.
UTM zone 51N	120deg to 126deg Northern hemisphere. China. Indonesia. North Korea. Philippines. Russia. South Korea. Taiwan.
UTM zone 51S	120deg to 126deg Southern hemisphere. Australia. East Timor. Indonesia.
UTM zone 52N	126deg to 132deg Northern hemisphere. China. Indonesia. Japan. North Korea. Russia. South Korea.
UTM zone 52S	126deg to 132deg Southern hemisphere. Australia. East Timor. Indonesia.
UTM zone 53N	132deg to 138deg Northern hemisphere. China. Japan. Russia.
UTM zone 53S	132deg to 138deg Southern hemisphere. Australia. Indonesia.
UTM zone 54N	138deg to 144deg Northern hemisphere. Japan. Russia.
UTM zone 54S	138deg to 144deg Southern hemisphere. Australia. Indonesia. Papua New Guinea.
UTM zone 55N	144deg to 150deg Northern hemisphere. Japan. Russia.
UTM zone 55S	144deg to 150deg Southern hemisphere. Australia. Papua New Guinea.
UTM zone 56N	150deg to 156deg Northern hemisphere. Russia.
UTM zone 56S	150deg to 156deg Southern hemisphere. Australia. Papua New Guinea.
UTM zone 57N	156deg to 162deg Northern hemisphere. Russia.
UTM zone 57S	156deg to 162deg Southern hemisphere.
UTM zone 58N	162deg to 168deg Northern hemisphere. Russia.
UTM zone 58S	162deg to 168deg Southern hemisphere.
UTM zone 59N	168deg to 174deg Northern hemisphere. Russia.
UTM zone 59S	168deg to 174deg Southern hemisphere. New Zealand.
UTM zone 60N	174deg to 180deg Northern hemisphere. Russia.
UTM zone 60S	174deg to 180deg Southern hemisphere. New Zealand.

The option **UTM-DEGREE-conversion** allows to convert UTM-coordinates into degrees. You must enter the correct utm-zone and the xy-coordinates will be automatically transformed into degrees or arcsecs.

The option **national conversion** allows to convert GPS degree traceheader coordinates to local national cartesian coordinates. At present the following national coordinates are supported:

RD-conversion (netherlands)

Swiss CH1903

Swiss CH1903+LV95

Swiss CH1903+LV03

These conversions Swiss CH1903+LV95 and Swiss CH1903+LV03 use the local reference system CH1903+ with its reference frame LV95. With the option Swiss CH1903+LV03 an additional transformation of the plane coordinates LV95 to LV03 will be done. In order to use these options the free ReframeComDll_x86.msi from SwissTopo must have been installed.

If the distancedimension = 'foot' the utm-conversion gives foot-values otherwise meter values. You must specify where the latitude is stored within the traceheader (latitude in degrees either in x- (shot x, rec x and CMP x-pos) or y-coord). The longitude is automatically vice versa in y- or x-coord.

General:

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

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

To be considered: The traceheader coordinates are stored either using a 32 bit floating format (Reflexw formats 16 bit integer and 32 bit floating point) or using a 64 bit double precision format (Reflexw formats new 16 bit integer and new 32 bit floating point) - see also chap. 1.5.4 Import format specification. If the coordinates have very high values with small changes, the data representation of the 32 bit floating format may not be good enough. In this case the Reflexw new format with the 64 bit double precision format must be used when importing the original data or a constant offset should be subtracted from the coordinates before storing them into the traceheaders.

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

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

Create(3D)Ensembles: Within this groupbox it is possible to subdivide the current profile into different lineparts. The ensemble-nr. within the traceheaders controls the size of each linepart.

With the option **equidistant** activated the current profile will be subdivided into different equidistant lineparts. The option **nr. of traces** defines the number of traces per linepart (ensemble).

With the options **distance markers**, **comment markers** and **constr. change markers** activated the current profile will be subdivided into different lineparts at the given corresponding marker positions. With constr. change markers active only those markers with the construction change comment will be used. Activate the option **create** to generate the lineparts (the Ensemble-Nr. within the traceheaders will be updated). This option allows you to redefine the individual ensembles (2D-lines) of a 3D-file or of a 2D-file consisting of different parts. One application for the second case is that some filter (e.g. background removal, migration, option lineparts activated) may use this subdivision.

close: terminates this window

1.9.6.1 traceheader tabella

A table containing the traceheader coordinates and the traceheader gain opens either when activating the option "file/Edit traceheader/TraceHeader Tabella" or when using the option update together with the updatetype tabella within the traceheader menu or after having imported a single shot line.

The table shows the traceheader coordinates (distance and the x, y, z-coordinates of the receivers and the shots), the time delay, the gain and the time collect. These values can be **manually changed**. To be considered for manual editing: distance value and rec.-x value should be equal for each trace.

It is also possible to change the data from a special column. First you must click on the header parameter using the right mouse button. Then a small window opens with different possibilities:

read from file: load the data for one header column from a 2 column ASCII-file. Each line of the ASCII-file consists of two columns. The first column contains the tracenummer and the second one the value of the traceheader header which shall be read in. A linear interpolation inside and extrapolation outside will be done at those positions where no data have been specified.

Example with two values for the header parameter time collect:

10 63359.00

20 63405.00

add.const. value: allows to add a constant value for all data of the currently chosen column.

multiply by value: allows to multiply by a constant value for all data of the currently chosen column.

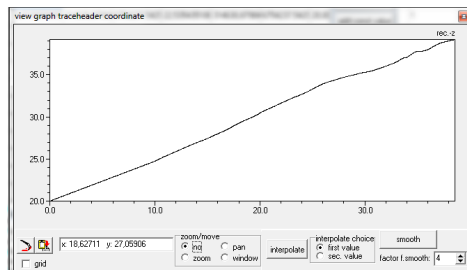
take over for all: The option allows to take over the actually set value for all values of the chosen column. For this purpose press the wanted cell using the left mouse button. Then click the right mouse button - the edit menu opens. Enter the new value without using any further mouse click. Click on take over for all.

Reverse order: allows to reverse order for a choosable number of traces. The number of traces must be even. The option might be useful if the order of a geophone array has been reversed during the data acquisition. In this case it might be useful to reverse the order both for the distances and for the x-rec position. Afterwards you may use the option resort-traceheader to be found under processing/trace interpolation in order to resort the data based on the new entered distances.

View graph: displays the data of the chosen column within a separate window where an interpolation and a

The screenshot shows a window titled "edit trace header coordinates for F:\data_project\geoscan\user\str\FOHDATA\DAT_0337.DAT". It contains a table with columns: trace-nr, distance, shot-x, shot-y, shot-z, rec-x, rec-y, time delay, gain, and time collect. The table lists 13 traces. Below the table are several interactive options: "topography/coordinates" (with sub-options like "3D-view of boreholes", "update shot pos.", etc.), "correct 1st offset", "interpolate", "smooth", "factor f smooth", "smooth shot coord", "source <-> rec", "interpolate all", "check distances", "load from file", "save on ASCII file", "save changes", and "close".

trace-nr	distance	shot-x	shot-y	shot-z	rec-x	rec-y	time delay	gain	time collect
1	0	449582	15051	6707713.3396	988.615	449582	std trace file	988.615	0
2	0.002188846	449582	15051	6707713.3396	988.62	449582	add const value	988.62	0
3	0.003772842	449582	15056	6707713.3396	988.625	449582	take over for all	988.625	0
4	0.139569335	449582	20846	6707713.3396	988.63	449582	view graph	988.63	0
5	0.201868039	449582	33594	6707713.4256	988.64	449582	close	988.64	0
6	0.268170145	449582	39142	6707713.4081	988.65	449582		988.65	0
7	0.311524426	449582	42835	6707713.4081	988.655	449582		988.655	0
8	0.354837910	449582	46336	6707713.5106	988.66	449582		988.66	0
9	0.398232801	449582	50233	6707713.5334	988.665	449582		988.665	0
10	0.441567246	449582	53265	6707713.5961	988.67	449582		988.67	0
11	0.504144016	449582	59811	6707713.6364	988.675	449582		988.675	0
12	0.558671363	449582	64116	6707713.6286	988.68	449582		988.68	0
13									



smoothing is possible. Click on **interpolation** and select the first and secondary point by a left or right mouse click within the graph. If zoom, pan or window is active (see below) the right mouse must be chosen. A linear interpolation between the two points will be done. With the option **shift** activated the values between the two points will be shifted by the entered amount instead. The option **smoothing** performs a running smooth of the data over an adjustable range of traces (option **factor f. smooth**). The data range may be **zoomed** and **panned**. With zoom activated you may increase or decrease the zoom in x- and y-direction by moving the mouse with pressed left mouse button. With pan activated you may move the data range. With window activated you may create a zoom window. If zooming is done the smoothing only concerns the zoomed data range.

The option **save changes**

saves the manual changes within the traceheaders of the current file.

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

The option **update from**

fileheader allows to update the receiver and shot-coordinates based on the current fileheader coordinates - active if entered from the import menu.

The option **update fileheader** allows to update the fileheader coordinates based on the traceheader coordinates. The option is useful in order to fit the fileheader and the traceheader coordinates - active if entered from the import menu.

The panel **fileheader coordinates** (only visible when entered from the import menu) shows the overall fileheader coordinates which can also be edited and allow an easy synchronization of the file and traceheader coordinates for the datatypes single shot, single shot/boreholes and single shot/VSP together with the options update from fileheader and update fileheader. With the option **update only shot coord.** activated only the shot coordinates will be updated when using the option update from fileheader. The option **non equidistant spread** allows to enter non equidistant receiver intervals within an additional column of the table. The first interval must be entered for trace-nr. 2, the rec.interval for trace-nr. 1 is always 0. After having entered these intervals together with the rec.start coordinate use the option update from fileheader in order to actualize the receiver coordinates. The distances will be automatically updated and the rec.end coordinate will be set to the last receiver value.

trace-nr.	distance	shot-x	shot-y	shot-z	rec.-x	rec.-y	rec.-z	time delay	gain	time collect
1	0	-26	49.0750000762	0	0	1	0	0	1	0
2	5	-26	49.0750000762	0	5	1	0	0	1	0
3	10	-26	49.0750000762	0	10	1	0	0	1	0
4	15	-26	49.0750000762	0	15	1	0	0	1	0
5	20	-26	49.0750000762	0	20	1	0	0	1	0
6	25	-26	49.0750000762	0	25	1	0	0	1	0
7	30	-26	49.0750000762	0	30	1	0	0	1	0
8	35	-26	49.0750000762	0	35	1	0	0	1	0
9	40	-26	49.0750000762	0	40	1	0	0	1	0
10	45	-26	49.0750000762	0	45	1	0	0	1	0
11	50	-26	49.0750000762	0	50	1	0	0	1	0
12	55	-26	49.0750000762	0	55	1	0	0	1	0

The option **save on AsciiFile** stores the traceheader coordinates on an ASCII-file with the extension dst. Each line of the ASCII file contains the following 8 informations:

tracenumber distance Shot-X-Pos Shot-Y-Pos receiver-X-Pos receiver-Y-Pos Shot Z receiver Z

The option **load from AsciiFile** allows to load the traceheader coordinates from an ASCII-file with the extension dst. Each line of the ASCII file contains the following 8 informations:

tracenumber distance Shot-X-Pos Shot-Y-Pos receiver-X-Pos receiver-Y-Pos Shot Z receiver Z

An automatic interpolation is done if for various traces no coordinate data are present within the ASCII-file.

If all distances within the ASCII-file are zero (2.colum) the distances will be calculated from the total difference between each shot and receiver positions (x,y and z-coordinate are taken into account).

The option **interpolate** allows to interpolate between two given coordinates. The interpolation must be done within one single traceheader coordinate group (for example only rec-z). There are two ways how to proceed:

- activate the option interpolate, then click on the wanted first coordinate and then on the second one. Inbetween a linear interpolation will be done.
- If you have changed manually any coordinate click on interpolate and then on the second coordinate. Again the linear interpolation will be performed.

With the option **interpolate all** activated all spreadsheet parameters will be interpolated for the chosen rows.

With the option view rec.geometry activated it is possible to select the coordinates within the view profile image using the left mouse button. Click on the first coordinate and then on the second one (both are indicated by a blue cross) and the interpolation will be done automatically between these two positions for all coordinates.

The option **source <-> rec.** allows to exchange the shot and receiver coordinates. The option might be useful if the shot position has been continuously changed with a fixed receiver. Such a geometry should be exchanged because the following interpretation methods all base on a fix shot position and changing receiver positions.

The option **x <-> y** allows to exchange the x- and y-coordinates of the sources and the receivers.

The option **rec -> source** allows to save the receiver coordinates within the shot coordinates.

The option **project on x-coord.** calculates the distances along the line based on the receiver coordinates (see also option calculate distances) and projects these distances onto the x-axis: the x-receiver coordinates are set to these distances and the y-coordinates are set to 0. The distance of the shot is also automatically determined and the shot x-coordinate is set to this distance. This is not strictly correct if the shot is situated within the receiver line. The start distance is queried after having activated the option.

The option does not work correctly if the profile consists of several shots with different receivers.

If the end-coordinates are smaller than the start coordinates an automatic flipping in profile direction is done before calculating the distances.

The option might be useful if GPS-coordinates are present within the traceheaders. These coordinates cannot be used for most of the refraction interpretation tools which need the coordinates only along the x-axis.

The option **topography (x,z values)** reads the topography (shot and receiver z-values) from an ASCII-file. Each line of the ASCII-file contains the distance and the depth or elevation. The option allows the independent input of the topography for the shot and the receiver coordinates (options update shot z-pos. and update receiver z-pos.). With activated option use x-traceheadercoord. the x-receiver traceheader coordinate (rec-x) is used and with deactivated option the distance within the traceheader is used. A linear interpolation or extrapolation is done where no values are defined within the ASCII-file.

With the option apply x-z topography activated the geometries of the shots and the receivers are recalculated based on the topographic xz-values. 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 option is identical to the option apply x-z topography within the travelttime analysis module with the difference that the z-values must be given as depths. It is assumed that the x-values of the data do not represent the correct x-coordinates but are determined directly on the topographic interface. The program automatically determines the positions of all receivers on the given topography and calculates the x- and z-projections of these positions. The x-coordinate within the ASCII-file represents the true x-coordinate within the xz-coordinate system. The following assumptions must hold true:

1. The topography file must contain the values for x=0 (fixed point) because this value serves at the starting point for determining the corrections. All corrections are relative to the value at x=0.
 2. The profiledirection is pure x, this means that the y-coordinates of all shots and receivers are equal.
 3. The original x-values are taken directly on the topographic interface.
- The application of this option is useful if you want to introduce a topography for the wavefront-inversion or for the raytracing (chap. 4.7).

With the option get distance along topography only the distance along the topography will be calculated and saved within the distance traceheader.

The option **check rec.coordinates** checks the receiver coordinates for breakouts and interpolates these. For that purpose the two x and y mean absolute differences between successive xy-receiver positions are calculated. If the absolute difference between two successive xy-receivers positions multiplied by the value of the option factor f.check exceeds one of the two mean difference value an interpolation for both the xy-shot and receiver coordinates is done for this xy-position. If the breakout extends over more than one trace again an interpolation is done. In this case the new coordinates will be automatically saved within the traceheaders.

The option **smooth rec. xy-coord.** allows to perform a running smooth of the xy-receiver coordinates over an adjustable range of traces (option factor f. smooth). The option might be useful for GPS-data especially if the option update distances

(see below) shall be used afterwards. After having completed the smoothing the original (red color) and the smoothed coordinates (black color) will be displayed for a direct check (see also option view rec. geometry). If the smoothing is not correct use the option reload from file and try again using a different smoothing parameter.

With the option **smooth shot coord.** activated the shot xy-coordinates will be smoothed in addition to the receiver coordinates.

Smooth rec- z-coord: allows to smooth the receivers (and optionally the shot) z-coordinates.

The option **correct lat.offset** allows to correct a lateral shot offset (shot-y) which is different from the receiver lateral offset (rec-y). As a result the receiver x-positions will be changed according to the true distances from the shot and the shot and receiver y-positions will be set to 0.

The option **view rec. geometry** allows to display the receiver coordinates within and an additional window (any curvature of the line coordinates is displayed). The option **use GPS map** for the option **view rec. geometry** - allows to display the geometry within a GPS Map.

With the option **free of distortion** activated the x- and y-axis are equally scaled for display. With the option **z-coord. as sec.coor.** activated the secondary coordinate is assumed to be the z-coordinate, if deactivated the y-coordinate is used. With the option **show shot positions** activated the shot positions stored within the traceheaders are displayed in addition in green (blue for 2.line) color. With the option **use lineparts** activated the coordinates lines will be displayed under consideration of the line parts (if the ensemble number changes the coordinates will not be combined using a line). With the option **show all coordinates** activated all coordinates will be plotted. Deactivating the option means that the coordinate line will be resampled based on the max. number of 1000 for the display. **Zooming** and **copy to clipboard** possibilities are included. The receiver coordinates selected within the spreadsheet will be displayed using a blue cross. If the option interpolate (see above) has been activated it is possible to select the coordinates within the view profile image (not within the Google Map) by using the left mouse button. Then the first row of the spreadsheet jumps on this position. If the option **interpolate all** is active in addition it is possible to select both the start and the end coordinate within the view profile image. Click on the first coordinate and then on the second one (both are indicated by a blue cross) and the interpolation will be done automatically between these two positions for all coordinates.

The **update distances** option allows to update the traceheader distances along the profil line based on the x- and y-receiver traceheader coordinates. If the option use z-coordinates is activated the z-receiver coordinates are also taken into account for the distance calculation. The start distance is taken from the fileheader. For the datatypes single shot and borehole.... the distances are taken from:

receiver x-coordinate for single shot

receiver y- or z-coordinate for borehole data depending on the setting of the option use z-coordinate.

The option is also available within traceheader menu (chap. 1.9.6). The option might be useful e.g. for non equally spaced seismic refraction data together with

activated plotoption TraceHeaderDistancies. This might be useful for example for the interactive use of the intercept time method for non equally spaced data.

The option **coordinate transformation** allows to perform the following coordinate transformations: UTM-conversion, UTM-degree conversion and plane projective transformation. The option **several files** allows to perform the coordinate transformation automatically for a choosable number of datasets. After having clicked on start the wanted datasets are queried.

The **plane projective transformation** allows the transformation of the original coordinates into another coordinate system using four pairs of transformation coordinates. This transformation allows you for example to use a local coordinate system and to transform these coordinates into UTM-coordinates (example shows data from a 24 channel GPR antenna array - in total 7 different arrays).

UtmZone

zone

☐ automatic

☐ distinct zone

☐ degree

☐ arcsec

latitude

☐ x-coord

☒ y-coord

coordinate transformation

TransformationRadioGroup1

☐ UTM-conversion

☐ UTM degree conversion

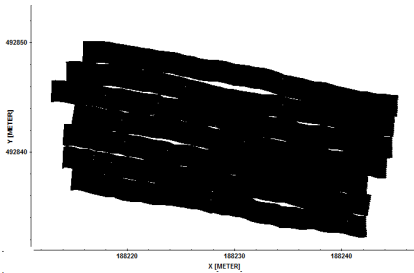
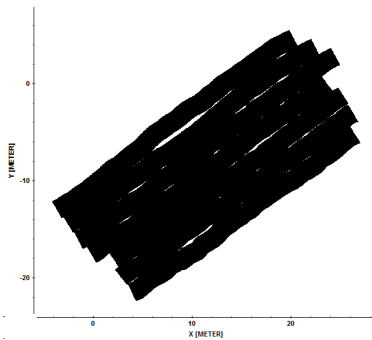
☒ plane projective transformation

plane projective transformation

x-in	y-in	x-out	y-out
-5.37	-12.32	188246.43	492845.58
6.42	-16.26	188237.7	492836.74
17.870001	-5.16	188222.47	492841.46
17.99	5.85	188217.38	492851.22

load 4 pairs of points

start



The option **apply borehole deviations** allows to apply the borehole coordinates which must have been stored within 2 different ASCII-files. Each line of the ASCII-file contains the XYZ-coordinates in meter. The Z values must be given as above sea level. The first line corresponds to the borehole top level. The original z-positions (see also chap. 6.2) are assumed to be relative to this top level and must be negative. The original xy shot and receiver coordinates will be ignored. If one of the borehole files is completely empty no changes will be done for this borehole.

After having pressed the button generate depth files the program asks the two borehole filenames (multiple filechoice). It is assumed that the original z-positions have been measured along the boreholes. The program recalculates these positions both for the shots (corresponds to borehole 1) and the receivers (corresponds to borehole 2) based on the given borehole data. A linear interpolation is done between the individual borehole coordinates. The following table shows the file structure of 2 borehole data and the pictures show the coordinates before and after the borehole correction. To be considered: The first receiver within the example corresponds to the greatest depth.

50000.0 45300.0 110.0	50005.0 45304.0 110.0
50000.1 45300.1 108.5	50005.1 45304.1 106.5
50000.2 45300.2 106.5	50005.0 45304.2 102.5
50000.3 45300.2 102.5	50004.9 45304.2 100.5
50000.4 45300.1 100.5	50004.7 45304.1 98.5
50000.3 45300.0 95.5	50004.4 45304.0 92.5
50000.2 45300.1 90.5	50004.2 45304.1 88.5
50000.1 45300.2 85.5	50004.1 45304.2 84.5
50000.1 45300.3 80.5	50004.1 45304.3 81.5

edit trace header coordinates

tracenr.	distance	shot-x	shot-y	shot-z	rec.-x	rec.-y	rec.-z	time delay	gain	time collect
1	0	0	0	-31	10	0	-31	0	1	0
2	0	0	0	-30	10	0	-30	0	1	0
3	0	0	0	-29	10	0	-29	0	1	0
4	0	0	0	-28	10	0	-28	0	1	0
5	0	0	0	-27	10	0	-27	0	1	0
6	0	0	0	-26	10	0	-26	0	1	0
7	0	0	0	-25	10	0	-25	0	1	0
8	0	0	0	-24	10	0	-24	0	1	0
9	0	0	0	-23	10	0	-23	0	1	0
10	0	0	0	-22	10	0	-22	0	1	0
11	0	0	0	-21	10	0	-21	0	1	0
12	0	0	0	-20	10	0	-20	0	1	0

CrossholeGroupBox
 apply borehole deviations
 3D-view of boreholes
 sample number: 4036
 trace number: 31

TimeDimension: ms
 start time: 0
 nom.frequency: 0
 get

source <-> rec.
 x <-> y
 check rec. coordinates
 factor f.check: 10
 smooth rec. xy-coord.
 factor f.smooth: 4
 interpolate

load from AscFile
 save on AscFile
 reload from file
 save changes
 close

edit trace header coordinates

tracenr.	distance	shot-x	shot-y	shot-z	rec.-x	rec.-y	rec.-z	time delay	gain	time collect
1	0	50000,1	45300,3	79,02488795	50004,1	45304,3	79,03905291	0	1	0
2	0	50000,1	45300,3	80,02488795	50004,1	45304,3	80,03905291	0	1	0
3	0	50000,1	45300,289504	81,024783007	50004,1	45304,3	81,03905291	0	1	0
4	0	50000,1	45300,269508	82,024583067	50004,1	45304,282041	82,038753693	0	1	0
5	0	50000,1	45300,249512	83,024383127	50004,1	45304,24872E	83,03819860C	0	1	0
6	0	50000,1	45300,22951E	84,024183187	50004,1	45304,215411	84,037643507	0	1	0
7	0	50000,1	45300,20952C	85,023983247	50004,1	45304,186573	85,037051162	0	1	0
8	0	50000,110473	45300,18952E	86,023678624	50004,13841C	45304,16158E	86,036426747	0	1	0
9	0	50000,130465	45300,169534	87,023278864	50004,16339E	45304,136604	87,035802332	0	1	0
10	0	50000,150457	45300,149542	88,022879103	50004,18837E	45304,11162C	88,03517791E	0	1	0
11	0	50000,170445	45300,12955C	89,022479342	50004,22670C	45304,08664E	89,034053687	0	1	0
12	0	50000,190441	45300,10955E	90,022079583	50004,276624	45304,061687	90,03249483E	0	1	0

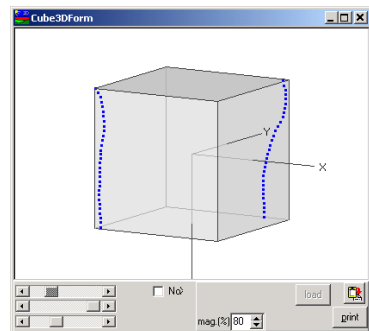
CrossholeGroupBox
 apply borehole deviations
 3D-view of boreholes
 sample number: 4036
 trace number: 31

TimeDimension: ms
 start time: 0
 nom.frequency: 935,0555
 get

source <-> rec.
 x <-> y
 check rec. coordinates
 factor f.check: 10
 smooth rec. xy-coord.
 factor f.smooth: 4
 interpolate

load from AscFile
 save on AscFile
 reload from file
 save changes
 close

The option **3D-view of boreholes** allows a 3D-view of the shot and receiver xyz-coordinates. The cube may be interactively rotated using the left mouse key.



1.10 Edit several FileHeaders

Within this menu you may change the fileheader coordinates and the start time (see also fileheader time specification.) of a choosable number of lines.

filename	dir	con	x-start	x-end	y-start	y-end	trace-inc	numb	time-inc	starttime	shot pos	S/R-distance	comment	comment	chann
DAT_0045.10T	X	Y	0	1778	0	0	1	0	0.1010100:0	0	0	6.500010	=128		0
DAT_0046.10T	X	Y	0	837	0	0	1	0	0.1010100:0	0	0	6.500010	=128		0
DAT_0047.10T	X	Y	0	898	0	0	1	0	0.1010100:0	0	0	6.500010	=128		0
DAT_0048.10T	X	Y	0	784	0	0	1	0	0.1010100:0	0	0	6.500010	=128		0
DAT_0049.10T	X	Y	0	510	0	0	1	0	0.1010100:0	0	0	6.500010	=128		0
DAT_0050.10T	Y	X	0	0	100	1293	1	0	0.1010100:0	0	0	6.500010	=128		0
DAT_0051.10T	X	Y	0	455	0	0	1	0	0.1010100:0	0	0	6.500010	=128		0

The following parameters can be changed within the table input:

ProfileDirection, ProfileConstant, XStart, XEnd, YStart, YEnd, trace-increment, Number, time-increment, start time, shot position, S/R-distance , 2 comments and the channel for multichannel data.

For single shot data the parameters are changed accordingly: receiver start and end, receiver offset, trace-increment, number, time-increment, start time, shot position and shot offset as well as 2 comments.

In addition the distance and time dimension may also be changed if desired (one single entry for all chosen profiles).

The wanted files are loaded within the **filechoice** groupbox:

Filefilter allows to enter a filter for the multi file choice.

Filepath specifies the path for loading the files to be processed.

OpenFiles opens the files for which the file header coordinates shall be edited. After having chosen the option you must select the wanted files. The fileheader coordinates are read in and shown within the table. It is only allowed to load data from one directory.

LoadTable loads a previously stored table from an ASCII-file. The file must have the extension TAB and must be stored under the path ASCII under the current project directory. To be considered: Within the ASCII file only the filenames of the individual lines without the filepath is stored. The filepath is automatically determined from the entered filepath and the current project directory.

SaveTable saves the current table on an ASCII-file. The file automatically receives the extension TAB and will be stored under the path ASCII under the current project directory. The file is ready for editing using any table calculation program like Excel or Quattro Pro.

ChangeProcLabel: allows to change the processing label for the chosen datafiles. A **new proc. label** must be entered. Already existing datafiles with this processing label will be removed.

The option within the **UpdateGroupBox** controls what is going to be changed within each fileheader.

The option **interpolate** allows to interpolate between two given values. The interpolation is only possible within one column (for example trace-inr.). There are two ways how to proceed:

a. activate the option interpolate, then click on the wanted first value and then on the second one. Inbetween a linear interpolation will be done.

b. If you have changed manually any value click on interpolate and then on the second coordinate. Again the linear interpolation will be performed.

The option may also be used in order to change all values to new single one.

FixTraceinrement: With this option activated the trace increment of each file will remain fixed. This means when changing the starting coordinate in profile direction the corresponding ending coordinate will be automatically changed based on this new coordinate and the fixed trace increment.

With this option deactivated the trace increment will be changed based on the set starting and ending coordinates.

UpdatePlotSettings: With this option activated the current plot settings are stored within each fileheader when activating the option save. The PlotOptions menu can be entered within this menu using the corresponding speed option.

UpdateAxisNames: With this option activated the manual set axis names are stored within the fileheader when activating the option save (see also Plotsuboptions - x- or y-axisname).

UpdateDistanceDimen.: With this option activated the currently entered distance dimension is stored in the fileheader of each profile.

UpdateTimeDimension: With this option activated the currently entered time dimension is stored in the fileheader of each profile.

Update Datatype: With this option activated the currently entered data type is stored in the fileheader of each profile.

UpdateValueDimension: With this option activated the currently entered value dimension is stored in the fileheader of each profile.

insert markers: insert markers at the first and last trace of each profile.

UpdateTraceHeaders: This radio box allows you to automatically update the geophone-, shot- and CMP-coordinates either based on the distance coordinates

of the file header (coordinates in profile direction and in profile constant) or based on an ASCII-file - see also option *actualize* within the trace header menu (chap. 1.9.6).

The option **offset -> x** is enabled for the **fileheader** update and for single shots. If activated the shot and receivers offsets are written to the corresponding x-traceheadercoordinates and the shot and receiver positions are written to the y-coordinates. If deactivated the y-traceheadercoordinates are used for the offsets and the x-coordinates are used for the positions.

With the option **ASCII-file** or **ASCII-file/int.** activated the DST files for the individual datafiles must exist. They must exist under the path ASCII under the project directory and the filename (without path and extension) must be the same like for the datafile and they must have the extension DST. Each line of the ASCII file contains the following 6 or 8 (z-pos. optional) informations:

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

Defining the trace number gives you the opportunity to read the geometry only for a distinct part of the data.

With the type **ASCII-file/int.** activated an automatic interpolation is done if for various traces no coordinate data are present within the ASCII-file. The option **distancies instead of tracenos** controls if the tracenumbers or the distances are used for the synchronization. With this option activated the distances defined in column 2 are used together with the distance coordinates of the file header in order to update the traceheader coordinates.

In addition the following **GPS-formats** are supported: RAMAC-GPS (*.cor and *.corc if corc-file is activated), WSKTRANS-GPS (*.utm,*tsp), CVS-GPS (*.cvs), IDS-GPS (*.gps), IDS-times (info????.txt), IDS-GEOX (*.geox), PulseEkko-GPS (*.gps and *.gp2), UTSI-GPS (*.gps), GSSI-GPS (*.dzt), NWI-GPS (*.nwi), GEODE-GPS(*.log), Marker-GPS(*.gps, only NMEA data file supported) and gps-times or gps-times spatial interpolation (*.gps, only NMEA data file supported or *.gpx if no gps-file is given or csv file from Proceq if present) (see also

Traceheader Edit, chap. 1.9.6 for a detailed description of the formats). For each format the ASCII-GPS datafiles must exist and must have the same name like the original datafile except the extension. For IDS-times format the filename info????,txt must include four last characters of the original datafilename. The option **latitude** (x-coord./y-coord.) specifies where the original latitude (or 1. coordinate in general) shall be stored - either on the x- or the y-position within the Reflexw traceheader. The option is available for all GPS-types. Setting the option to y-coord. might be useful if you are using for example the option *view/profile* line in order to have the same map orientation as in normal geographic maps.

The actual setting is saved when leaving the program. The option **NMEA-string no degree** allows if activated that the \$GP Strings do not contain degree values but the string will be interpreted as it is.

Pecularity for IDS-GPS data: The 4th character indicating the channel number within the automatically defined GPS-filename will be set to 1 if the corresponding filename does not exist. This might be necessary when multiple channels of data were collected. The option **1. orig. coord.** allows you to specify where the first original coordinates shall be stored when importing GPS-data - either on the x-traceheader coordinates or on the y-traceheader coordinates.

With the option **read from one file** (only for WSKTRANS-GPS, gps-times and gps-times spatial interpol.) activated one single gps-file will be queried and used for all chosen original datafiles. The option might be useful for the simultaneous import of the data of one antenna array with one single GPS-unit and using the

option **correct for offset** together with activated option **offsets from file** for a correction of the gps-coordinates based on the array configuration. The option is also available within the import menu.

Trace number based GPS-synchronization like Ramac-GPS, IDS-GPS, GSSI-GPS, UTSI-GPS and PulseEKKO GPS:

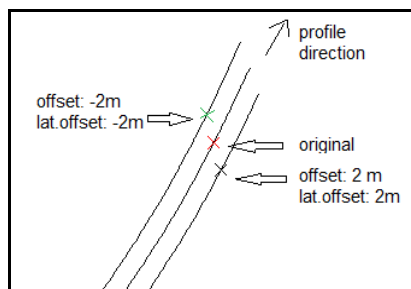
now the **original trace numbers** are used for the synchronization of the GPS coordinates if a format was used in which the synchronization is based on the trace numbers. This allows processing steps that change the sorting or the number of original trace numbers (e.g. x-flipping in profile direction if the option **meandering** has been activated) to be carried out before the GPS coordinates are updated. With the option **use act trace no** activated the actual Reflexw tracenumbers are used instead of the original ones. This might be useful if the gps-datafile has been created afterwards based on the new tracenumbers after any processing step, e.g. an x-flipping.

Activating the option **UTM-conversion** allows to convert GPS degree traceheader coordinates to local cartesian coordinates using the so called Universal-Transverse-Mercator(UTM)-conversion (WGS84, see also chap. 1.9.5). The option **UTMtoDegree** allows to convert UTM-coordinates into degrees. You must enter the correct utm-zone and the xy-coordinates will be automatically transformed into degrees. The option **calculate distances** allows to update the traceheader distances based on the x- and y-receiver traceheader coordinates. The start distance is always set to 0. The options **use z-coordinates** and **straight line from start point** defined within the edit traceheader menu are valid (see chap. 1.9.6).

Activating **smooth xy-coord.** and/or **smooth z-coord.** allows to smooth the traceheader receiver coordinates over a selectable number of traces. For common offset lines the same smoothing will be done for the shot coordinates.

The option **correct for offset in profile direction** or activating the option **correct for offset** (e.g. in addition to a GPS-cimport) allows to correct the xy-GPS coordinates for an offset in profile direction (option **offset**) and perpendicular to it (option **lateral offset**). The option might be useful if there exists any offset between the geophones/shots and the GPS-unit, e.g. a data acquisition on a ship with a boomer and hydrophones. The **offset** in

profile direction will be subtracted from the distances in profile direction (this means a negative value must be entered if the GPS unit is located behind the receivers in profile direction and a positive value must be entered if the GPS unit is located before). A positive **lateral offset** results in a shift to the right of the profile direction, a negative lateral offset results in a shift to the left. If the distances within the traceheader are specified and the lateral offset is set to 0 the correction is done based on these trace header distances (see also update option **calculate distances**). This means that the corrected values follow exactly the track of the GPS-unit. If an extrapolation is necessary at the end or at the



start of the profile (depending on the correct value) the program calculates the actual direction of the profile from the difference of the actual GPS-coordinate and the previous one. It is possible to select a **smooth** range for the definition of the profile direction. If no distances are stored within the traceheaders or if the lateral offset is not 0 this procedure will be applied for all traces.

The option **offsets from header** allows to read the correct offsets offset and lat.offset from the fileheader of each file. Offset will be set to the fileheader startcoordinate in profile direction and lat. offset will be set to the fileheader startcoordinate in profile constant direction.

The option **offsets from file** allows to read the correct offsets (in profile direction - offset - and perpendicular to it - lat.offset) from an ASCII-file named CorrectOffset.txt which must have been stored under the subdirectory ASCII under the actual project directory. Each line of the file must contain the following parameters:

"channel number" "offset" "lateral offset"

Example of 3 channels with a symmetrical array:

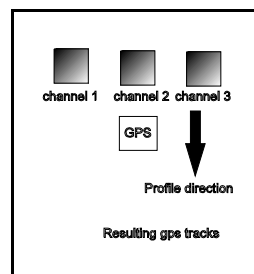
1 0.5 0.5

2 0.5 0.0

3 0.5 -0.5

The **offset** in profile direction will be subtracted from the distances in profile direction (this means a negative value must be entered if the GPS unit is located behind the receivers in profile direction and a positive value must be entered if the GPS unit is located before). A positive **lateral offset** results in a shift to the right of the profile direction, a negative lateral offset results in a shift to the left. **Channel number** defines the channel number of a multichannel device. The number may be any number between 1 and 64. The option has been introduced in order to automatically correct the gps-coordinates for a multichannel system (up to 64 channels). The corresponding channelnumber of the datafile has been taken from the original filename during the import and stored within the fileheader. This is in general the last two characters of the filename (exception for IDS-data: here the third and forth digits are used for the definition of the channel number). For GSSI-data the channelnumber is automatically determined from the multiplexed order of the original data.

To be considered: the z-coordinates will also be updated if no interpolation is necessary, this means the correction is only done in profile direction (lateral offset 0 and smooth direction set to back). Precondition is that the traceheader distances have been defined before (option calculate distances) or by activating the option **use fileheader dist.** which uses the fileheader distances instead of the traceheader distances for the pure correction in profile direction.



Pecularity for single shot data:

correct for offset: if the datatype of the loaded profiles is single shot the correction is done based on the option **correct lat.offset** within the edit traceheader tabella. The option allows to correct a lateral shot offset (shot-y) which is different from the receiver lateral offset (rec-y). As a result the receiver x-positions will be changed according to the true distances from the shot and the shot and receiver y-positions will be set to 0.

The option **latency** allows to correct a possible time delay between the gpr and the gps times. If this time delay has not been considered during the synchronization of both devices it may be corrected manually using this option. The entered latency in s will be subtracted from the times stored within the Reflexw traceheaders and this new value is the base for searching the corresponding gps coordinates which replace the old ones. Normally the latency will be given in positive values and thereby at the beginning of the gpr line no gps coordinates are present. Here an automatic extrapolation will be done.

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

Using the option **interpolation equal data** controls if identical coordinates for the x- and y- shot and receiver positions for subsequent traces are present and performs an interpolation for those traces.

The option **exchange xy-coordinates** allows to exchange the x- and y-shot and receiver coordinates.

UpdateTable: update the table shown above based on the new set parameters. This means the ending coordinates are recalculated if FixTraceincrement is set to true.

old format: if activated the parameter files will be saved using the old format (up to version 7.5).

save: save the current parameter settings within each fileheader

print: prints the current table

MarkerRelocate: allows the relocation of the start- and end-profile-direction coordinate based on the marker defined by the parameter **number** and the preset **position**. The program searches the number. marker within each line and sets its position to the preset position value. The start and end coordinates in profile direction are automatically updated (see also fileheader ControlPanel.). With the option **comment marker** activated the comment markers are loaded in addition to the distance markers (see also edit comment markers).

1.11 Processing MenuItem

The data processing is completely interactive. With the option Apply on example trace activated the effect of the filter is online controlled when changing the filter parameters by showing both the original and filtered trace. A great variety of one and multi channel filters as well as edit and trace interpolation functions are included. All edit inputs may be entered either interactively in the original profile or using the table input.

Each processing step can only be applied on the primary (1.) line. In order to apply any processing step on the secondary (2.) line first you must define the secondary line as the primary line using the option ChangeSecondToPrimary.

The data processing facilities are divided into different groups:

- 1D-Filter processing

 - Gain

 - StaticCorrection/muting

 - Decipping/arithmetic functions

 - Deconvolution/shaping

 - complex traceanalysis/spectral analysis

 - Trace Interpolation/Resorting

 - Edit traces/traceranges

- 2D-Filter

 - migration/timedepth conversion

 - Fk-filter/data fusion

The sequence (batch) processing (see also Sequence Processing) facilitates a completely automatic sequence of processing steps for a choosable number of profiles. Primary and secondary section are displayed simultaneously, whereby a direct control of the result is given. You may choose between the so-called sequence mode and the single processing mode with the possibility of applying the processing steps individually on the primary profile.

1.11.1 1D-Filter

The following filter possibilities are included in the 1D-filter group:

Meanfilter
Medianfilter
bandpassfrequency
Bandpassbutterworth
filter/timedependent
notchfilter/frequ.
spectrum spikes
subtract-mean(dewow)
subtract-DC-shift
crosscorrelation
auto correlation
resampling
walsh bandpass

For all filters available within this group the original trace is shown together with the currently filtered trace, the spectrum of the original trace and the spectrum of the currently filtered trace. With the filter crosscorrelation activated additionally the reference trace is shown. This reference trace may also be saved for a subsequent viewing or preprocessing.

The arrows near the original trace and original spectrum are used to enhance the resolution of the trace and spectrum windows. The trace or the spectrum is stretched up and down when using these arrows. The spectrum scale is Hz, if the time scale is in milli-seconds and MHz, if the time scale is in nano-seconds, respectively.

In addition to the zooming arrows there exist an equivalent spin button which allows to move the starting point of the primary and filtered traces as well as of the corresponding spectra.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number** or by clicking in the primary file using the left mouse button.

The filter parameters for the individual processing steps are entered within the **filter parameter group**. The up/down arrows near the input fields allow to change the actual entered value by + or - 10 %. With the option **Apply on example trace** activated the effect of the filter parameters is online controlled when changing the parameters by showing both the original and filtered trace. The arrows near the input fields may be best used for such an interactive control. Deactivate this parameter if this control takes too much computer time.

After having specified the filter parameter, you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing.

Activating the option **sequence proc.** additionally open the Sequence Processing menu. With this menu opened you may easily add the current

processing step to the sequence processing flow which may be applied to a choosable number of lines after.

Activating the option **close after processing** allows to automatically close the processing menu after having done the processing.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing. Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing.

1.11.1.1 Meanfilter

The filter acts on each trace independently. The filter performs a mean over a selectable number of time sample for each time step.

The filter parameter **mean range** determines the window width in samples over which an averaging of the amplitudes of a trace is performed. With a window width of 4 samples the mean value of the current, the two preceding and the two following samples of a trace is calculated and assigned to the current sample as a new value. If between the current sample and the beginning or the end of a trace lie fewer samples than half of the window width the window width is decreased on one side.

The effect of this mean value averaging complies with a lowpass filtering in the time domain, which is the more pronounced the larger the chosen window width. This filter is much faster than an application of a lowpass filter with, e.g. a bandpass filter, but not as variable.

1.11.1.2 Medianfilter

The filter acts on each trace independently. The filter performs a median over a selectable number of time sample for each time step.

The filter parameter **median range** determines the window width in samples over which the median of the amplitudes of a trace is performed. With a window width of 4 samples the current sample, the two preceding and the two following samples of a trace are sorted according to their value. The median of these samples, i.e. the third largest or the third smallest value, respectively, is assigned then to the current sample. If between the current sample and the beginning or the end of a trace lie fewer samples than half of the window width the window width is decreased on one side.

With the parameter **min. slope (%)** the samples for which the median filter shall be applied can be restricted. First the max. absolute difference between successive samples within the complete profile will be calculated. Then the median filter will only be applied if the absolute difference for the actual sample is larger than the percentage (given by the parameter min. slope (%)) of this max. difference. Specifying this parameter allows you for example to only remove spikes (which exhibit very large absolute differences) without influencing the rest of the data.

The median filter can be employed for deleting spikes from the data. The lowpass effect of the median filter is the stronger the larger the selected window width.

1.11.1.3 Bandpassfrequency

The filter acts on each trace independently. Here you can apply a bandpass filter to each trace. The filter works within the frequency domain.

Four different types of bandpass filtering are available: `cos-taper`, `cos**2-taper`, `butterworth` and `high/low butter.`

The filter band for the types **`cos-taper`** and **`cos**2-taper`** is specified by the setting of four frequency values. The first point determines the **low-cut frequency**, the second one the beginning of the plateau (**lower plateau**). Between the low-cut frequency and the beginning of the plateau the filter is represented by a cosine-window or a cosine**2-window respectively. The third point determines the end of the plateau (**upper plateau**) and the fourth the **high cut frequency**. Between these points the filter is represented by a cosine-window or a cosine**2-window, too. The frequency spectrum below the low cut and above the high cut frequency are set to zero. By the corresponding choice of the points of the bandpass either a lowpass or a highpass can be approximately realized. The cosine-range is used to suppress unwanted reverberations resulting from the filter operator. The cosine range should be adapted to the spectral amplitudes existing at the high and low-cut frequencies. If the spectral amplitudes are quite low, the cosine-range can be decreased.

The filterband for the types **`butterworth`** and **`high/low butter.`** is specified by the setting of the two frequency values **lower cutoff** and **upper cutoff**. The description of the filter is given in Kanasevitch (1981). The type `butterworth` uses transformed frequencies which will be used for the definition of the transfer function of the bandpass. The type `high/low butter.` applies both a highpass and a lowpass butterworth filter on the data within one step in order to simulate a bandpass. The parameter **order** specifies the order of the butterworth filter. The higher the order the higher is the roll-off of the transfer function which on the other leads to larger reverberations. Therefore a good compromise must be found between reverberations and wanted passband.

The frequency values can be set either manually or within the frequency spectrum of the original example trace. For that activate the wanted filter parameter by simply enter it and then choose the wanted frequency value using the left mouse button within the frequency spectrum. The activated filter parameters is automatically updated.

With the option **zero padding** activated zeros will be automatically included during the bandpassfiltering in order to prevent aliasing in time domain. The option might be useful if a strong first arrival is visible.

Noise can be suppressed with the bandpass filter when it differs from the signal in its frequency content. It is recommended to select a trace containing both a significant portion of noise and of the signal for the filter parameters setting.

1.11.1.4 Bandpassbutterworth

The filter acts on each trace independently. Here you can apply a **bandpass filtering** in the time domain using a recursive filter.

The filter band is specified by the setting of two frequency values. The first point determines the **low-cut frequency**, the second one the **high cut frequency**. The frequency spectrum below the low cut and above the high cut frequency is set to zero. By the corresponding choice of the points of the bandpass either a lowpass or a highpass can be approximately realized.

The frequency values can be set either manually or within the frequency spectrum of the original example trace. For that activate the wanted filter parameter by simply enter it and then choose the wanted frequency value using the left mouse button within the frequency spectrum. The activated filter parameters is automatically updated.

Noise can be suppressed with the bandpass filter when it differs from the signal in its frequency content. It is recommended to select a trace containing both a significant portion of noise and of the signal for the filter parameters setting.

1.11.1.5 Filter/timedependent

The filter acts on each trace independently. Here you can apply a timedependent bandpass filtering in the time domain using a recursive filter.

The filter band is specified by the setting of two frequency value pairs. The first frequency value pair is valid for the time-begin and the second frequency value pair is valid for the time-end of each trace. The first point of each pair determines the **low-cut frequency**, the second one of each pair the **high cut frequency**. The frequency spectrum below the low cut and above the high cut frequency is set to zero. New filter boundaries are calculated for each time value based on the input value pairs (linear interpolation). Butterworth filtering is done using these calculated filter boundaries.

Example: you want to increase the lowpass filter effect with time because of stronger high frequency noise for greater times:
set the first filter pair resulting in a large bandpass (e.g. f1=100 MHz and f2=500 MHz).
set the second filter pair resulting in a smaller bandpass (e.g. f1=100 MHz and f2=300 MHz).
Using these values the high frequency content of the filtered data for smaller times is maintained whereas for greater times this frequency content is filtered out.

Corresponding to the bandpassbutterworth filter, the frequency values can be set either manually or within the frequency spectrum of the original example trace. For that activate the wanted filter parameter by simply enter it and then choose the wanted frequency value using the left mouse button within the frequency spectrum. The activated filter parameters is automatically updated.

1.11.1.6 Notchfilter/frequ.

The filter acts on each trace independently. Here you can apply a notchfilter to each trace in the frequency domain.

The filter band is specified by the setting of four frequency values. The first point determines the end of the low frequency plateau (**lower plateau**), the second one the **low-cut frequency**. Between the end of the low frequency plateau and the low-cut frequency the filter is represented by a cosine-window. The third point determines the **high cut frequency** and the fourth one the beginning of the high frequency plateau (**upper plateau**). Between these points the filter is represented by a cosine-window, too. The frequency spectrum between the cut frequencies is set to zero.

The frequency values can be set either manually or within the frequency spectrum of the original example trace. For that activate the wanted filter parameter by simply enter it and then choose the wanted frequency value using the left mouse button within the frequency spectrum. The activated filter parameters is automatically updated.

The cosine-range is used to suppress unwanted reverberations resulting from the filter operator. The cosine range should be adapted to the spectral amplitudes existing at the high and low-cut frequencies. If the spectral amplitudes are quite low, the cosine-range can be decreased.

A notch filter is especially suitable for suppressing nearly mono-frequency noise. For this you should select a trace containing a significant portion of noise. This facilitates the setting of the filter band.

1.11.1.7 spectrum spikes

The filter acts on each trace independently. Here you can apply a notchfilter to each trace in the frequency domain whereby the filterrange will be automatically determined from spikes within the frequency spectrum of the individual trace. The spikes are not determined the quotient or from the difference of the actual trace spectrum and the overall profile spectrum. Therefore this filter does not eliminate spikes which occur for each trace (e.g. 50 Hz noise) but only those spikes which only occur for special traces, e.g. due to guiding waves or some other resonance effects.

The parameter **max. nr. of spikes** defines the max. number of spikes within the frequency spectrum which will be automatically detected and filtered out. The parameter **spike threshold (%)** defines the threshold in percentage of the mean spectrum value for finding the spikes. The parameter **filter width** defines the range around the spike which will be filtered out.

The parameterbox **calculate type** allows to specify how the spikes are determined. With **normalization** activated a normalized quotient will be used. With **no normalizat.** activated the difference will be used.

The spectrum spikes filter is especially suitable for suppressing nearly mono-frequency noise which is not restricted to a special frequency but may vary from

trace to trace. Those traces for which no spikes have been detected no filtering will be done.

1.11.1.8 Subtract-mean (dewow)

The filter acts on each trace independently. With this option activated a running mean value is calculated for each value of each trace. This running mean is subtracted from the central point.

As filter parameter the **time window** for the calculation of the running mean value must be entered.

This filter may be used for eliminating a possible low frequency part (dewow). For this purpose the window range should be set to about one principal period.

1.11.1.9 Subtract-DC-shift

The filter acts on each trace independently. This option allows a so-called zero mean, i.e. the subtraction of an existing time constant shift.

As filter parameter two time values (**1.time** and **2.time**) must be entered. Within this time range the mean is calculated for each trace which is subsequently subtracted from all samples of each trace. Therefore it has to be guaranteed that the mean value in the corresponding time range complies with the shift you want to eliminate.

1.11.1.10 Crosscorrelation

This option allows to calculate the crosscorrelation of a time trace to be specified with all the other traces of the profile.

The trace used for crosscorrelation is chosen by the selection of a trace/time window given by the filter parameters. This trace is shown in the reference trace window. The first filter parameter specifies the start time (**start ref.time**) and the second one the end time (**end ref.time**). The third parameter **reference trace** gives the number of the trace used for the crosscorrelation. The fourth parameter **max. time** specifies the resulting time range which is for vibration data normally much smaller than the original time.

The switch groupbox allows you to specify whether the only one reference trace is specified within the **whole profile** or whether individual data parts are automatically determined for **line parts** separated by each other by a new ensemble number. In this case the reference trace gives the number within each ensemble, e.g. a multi shot datafile.

With the option **keep reference trace** activated the reference trace(s) for the crosscorrelation will be kept within the filtered datafile. Otherwise the filtered file does not contain the reference trace(s) any more.

The crosscorrelation works either within the frequency domain or time domain (option **work in frequency domain** active or deactivated). The CPU-time for the frequency domain filtering is much smaller than for the timedomain calculation.

The crosscorrelation can be used to improve the signal-to-noise ratio. Another effect is the shortening of the original wave train, if the conditions prevail (if possible a broadband spectrum).

Another important application is given if a **sweep** signal has been used.

1.11.1.11 auto correlation

This option allows to calculate the autocorrelation of each trace for a given time range.

Enter the timerange (options start time and end time) for which the autocorrelation will be generated. The max. value of the autocorrelation for each trace is set to the scaling factor.

1.11.1.12 Resampling

The filter allows you a resampling in time direction. The resampling is performed within the time or frequency range.

The filter parameter **timeincr.** determines the new sampling interval in time direction in the given time dimension.

The groupbox **working range** specifies whether the resampling is done within the timerange or within the frequency range. When using the timerange a linear interpolation is done between the datapoints. Using the frequencyrange the individual shape of the wavelets is taken into account more accurately. In this case the new timeincrement must be a multiple of the original one or must be divisible by it (e.g. original timeincrement: 0.5 - new time increment either 1.0 or 0.25).

The filter may be used for an equal redefinition of the sample interval of different 2D-lines which have been acquired with different time increments. Some processing steps like the combination of 2D-lines into a 3D-file require equal time increments. To be considered: decreasing the original time increment does not mean an increase of the vertical resolution.

1.11.1.13 Walsh bandpass

The filter acts on each trace independently. Here you can apply a bandpass filter to each trace using the Walsh transformation. The Walsh transformation is based on an orthonormal system of step functions instead of sin-functions which are used for the Fourier transformation. Therefore this filter is more suitable for stepwise data (e.g. timeslices) than for sinoidal data like the "normal" timeseries of wavedata.

The filter band is specified by the setting of the **lower cutoff** and the **upper cutoff** walsh coefficients. The max. Walsh coefficient is equal half the sample number. The walsh coefficients are not directly connected to the frequencies of the signal but in general one can say: the lower walsh coefficients represent more the lower frequencies whereas the higher coefficients are dominated by the higher frequencies. Therefore a lowpass filter is set by a small value of the upper cutoff parameter whereas a highpass filter is set by a high value of the lower cutoff parameter. A taper can be defined both for the lower and the upper cutoff coefficients (parameter **lower taper** and **upper taper**).

1.11.2 Gain

The following filter possibilities for changing the gain are included in the gain group:

- AGC-Gain
- energy decay
- gain function
- remove header gain
- div.compensation
- manual gain (y)
- manual gain (x)
- scaled windowgain(x)
- trace normalize
- x-distance decay(db)
- compensate stripes
- normalize profiles
- normalize 3D-file

For some filters available within this group the original trace is shown together with the currently filtered trace, the spectrum of the original trace and the spectrum of the currently filtered trace. With the filter energy decay activated additionally the reference trace is shown. This reference trace may also be saved for a subsequent viewing or preprocessing.

The arrows near the original trace and original spectrum are used to enhance the resolution of the trace and spectrum windows. The trace or the spectrum is stretched up and down when using these arrows. The spectrum scale is Hz, if the time scale is in milli-seconds and MHz, if the time scale is in nano-seconds, respectively.

In addition to the zooming arrows there exist an equivalent spin button which allows to move the starting point of the primary and filtered traces as well as of the corresponding spectra.

When activating the options manual gain (y) or manual gain(x) a table appears. With the option manual gain (x) activated the inputs may be entered either interactively within the original profile or using the table input. With the option manual gain (y) activated a new window appears which allows the interactive input of the new gain curve again with the possibility of entering the values within the table. The options save and load allow to store the current edit values to an ASCII file and to load the wanted edit values from file, respectively.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number**.

The filter parameters for the individual processing steps are entered within the **filter parameter group**. The up/down arrows near the input fields allow to change the actual entered value by + or - 10 %. With the option **Apply on example trace** activated the effect of the filter parameters is online controlled when changing the parameters by showing both the original and filtered trace. The arrows near the input fields may be best used for such an interactive control. Deactivate this parameter if this control takes too much computer time.

After having specified the filter parameter, you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing.

Activating the option **sequence proc.** additionally open the Sequence Processing menu. With this menu opened you may easily add the current processing step to the sequence processing flow which may be applied to a choosable number of lines after.

Within the **distance range** panel you may restrict the traces for which the filter will be applied. With **all traces** activated all traces within the profile will be taken into account. With deactivated option you may enter the **1. and the last trace** for applying the filter.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing. Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing.

1.11.2.1 AGC-Gain

The filter acts on each trace independently. The gain filter facilitates the creation of equally distributed amplitudes in y-direction (normally time axis) within a predefinable window. The information of the true amplitude is lost, of course.

The filter parameter **window length** determines the window size in the given timedimension. The program calculates at first an average amplitude over the total time range for each trace. After that the program scales each amplitude value in that way that the mean amplitude has the same value for each selected window around the current value within a trace. The size of the window determines the kind of amplitude equality distribution. A window size of only one sample increment means that each time sample within one trace receives the same amplitude value (no reasonable choice), a window size of the whole trace length causes no modification of the amplitude. Small window sizes cause a strong equality distribution, large windows a weak.

After the application of the AGC all data points are automatically multiplied by a **scaling factor**. You should choose this factor in such a way that all amplitude values lie within the 16 data bits if your REFLEX format is 16 bit Integer. If not, those amplitudes are automatically restricted to values of 32000 or -32000 in this case. If your REFLEX format is 32 bit floating point, the scaling value can be set to 1.

The option **max. gain** can be used in order to restrict the gain for each sample.

The option **normalize** performs a trace normalization. If this option is active the **max. normalize** value must be entered. This option is by default set to 2048.

Each value can be entered but if the data had been stored using one of the 16 bit integer formats the max. value should be within the 16 bit integer range and should not be less than 2048.

This option serves for emphasizing of low amplitude ranges against ranges with high amplitudes.

1.11.2.2 Energy decay

The filter acts on each trace independently. By activating this option a gaincurve in y-(time-) direction is applied on the complete profile based on a mean amplitude decay curve which is automatically determined. The information of the true amplitude is lost, of course.

First a mean decay curve is determined from all existing traces. After the application of a median filter on this curve every data point of each trace is divided by the values of the decay curve.

After the multiplication of the energy decay curve all data points are automatically multiplied by a **scaling factor**. You should choose this factor in such a way that all amplitude values lie within the 16 data bits if your REFLEX format is 16 bit Integer. If not, those amplitudes are automatically restricted to values of 32000 or -32000 in this case. If your REFLEX format is 32 bit floating point, the scaling value can be set to 1.

Analogous to the gain filter AGC-Gain this option serves for emphasizing of low amplitude ranges against ranges with high amplitudes.

1.11.2.3 Gain function

The filter acts on each trace independently. The filter facilitates the possibility of multiplying the data points by a given function $g(y)$ or $g(t)$ respectively.

The function $g(t)$ consists of a linear and an exponential part: $g(t)=(1+a*t)*e^{(b*t)}$ with $a=a'/\text{pulse width}$ and $b=b'*v/8.69$ with $v=0.1 \text{ m/nsec}$ or 1.0 m/msec respectively. The pulse width is automatically taken from the nominal frequency if given (see option FileHeader Edit). Otherwise, the nominal frequency is automatically determined from the first arrival. The two filter parameters a' (**linear gain**) and b' (**exponent**) must be entered. a' : not dimension except for the case of non set nominal frequency; b' : input in dB/meter. In addition you have to enter the **start time** (the filter starts at that time with value 1) and the **max. gain** which specifies the max. gain factor which will be multiplied with the data.

With the option "Apply on example trace" activated the resulting gain curve is plotted within the image next to the filtered trace.

The data are multiplied by this function in order to compensate for possible damping or geometric spreading losses.

1.11.2.4 Remove header gain

The filter acts on each trace independently. The option allows to apply the inverse of a digitized gaincurve stored in the fileheader.

The gainvalues used for the GSSI data-acquisition, which are stored in the RADAN-header, are automatically read in (see also option FileHeader Edit). Therefore it is possible to remove the gaincurve and to get raw data without timevariant amplification. Any gaincurve, which is applied on the data after the conversion (e.g. using the option manual gain (y)) is not automatically transferred into the fileheader. This must be done manually within the option FileHeader Edit (max. 8 values (time and gain)).

The filter is useful if you want to compare data files acquired with different gain functions or if you want to use the true amplitude information.

1.11.2.5 div. compensation

The filter acts on each trace independently. The option allows to compensate for the geometrical divergence losses.

The processing step applies a compensation of the geometrical divergence losses either proportional t (spherical losses - 3D-point source) or proportional \sqrt{t} (2-dimensional losses - line source).

1.11.2.6 Manual gain (y)

The filter acts on each trace independently. The option allows to interactively define a digitized gain curve in y-direction (normally time axis) and to apply this gain curve on the data.

After having activated the option a table and a new window appears which both allow the interactive input of the new gain curve (**time** in the given time dimension and **factor** in db). The chosen values are automatically transferred to the table input and vice versa to the gain window. Inputs between -20 and 80 db are possible. Between the individual marks the values are linearly interpolated, at the y-margins the specified value belonging to the smallest or largest y-coordinate, respectively, is extrapolated. The effect of the entered gain values is automatically shown within the filtered trace window.

The option **save** stores the current edit values on an ASCII file under the project directory with the name ygain and a freely choosable extension. Each line of the ASCII file consists of two columns: 1. column: time in the given time dimension, 2. column: factor in db.

The option **load** loads the wanted edit values from an ASCII file with the filename ygain. The table and the interactive window are automatically updated.

The option **reset** resets the current edit values.

The gainvalues entered are automatically transferred into the fileheader (max. 8 values (time and gain) - see also option FileHeader Edit).

The first and last gainvalue are stored within the processing flow.

The option may serve to emphasize y-ranges, where for example structures are assumed.

1.11.2.7 Manual gain (x)

The filter acts on each trace independently. The option allows to interactively define a digitized gain curve in x-direction (normally distance axis) and to apply this gain curve on the data.

After having activated the option a table appears which allows the interactive input of the new gain curve. The **distance** and the multiplication **factor** (linear) must be entered. Between the individual marks the values are linearly interpolated, at the x-margins the specified value belonging to the smallest or largest x-coordinate, respectively, is extrapolated. The distance can also be defined within the original profile by pressing the left mouse button at the desired position. The multiplication factor must be entered manually within the table.

The option **get factors for 3D-file** allows to automatically generate scaling factors for the individual 2D-ensembles of a 3D-file. The options start window and end window define the time range for calculating the scaling factors. The option might be useful to compensate different coupling conditions during the acquisition of a 3D-file.

The option **save** stores the current edit values on an ASCII file under the project directory with the name xgain and a freely choosable extension. Each line of the ASCII file consists of two columns: 1. column: distance in the given distance dimension, 2. column: factor.

The option **load** loads the wanted edit values from an ASCII file with the filename xgain. The table and the interactive window are automatically updated.

The option **reset** resets the current edit values.

The option serves to amplify profile areas, which are pronounced weaker than other areas because of higher attenuation for example.

1.11.2.8 Scaled windowgain(x)

The filter acts on each trace independently. The gain filter facilitates the amplitude equality distribution in x-direction for a y-window to be specified. For each trace one normalization factor is determined which will be multiplied by each value of the trace. Therefore the energy distribution within the trace will not be affected but energy differences inbetween the traces will be equalized.

After selecting the option the program demands the input of the time window (**start window** and **end window**). The filter facilitates a normalization of the amplitude sum within this window.

It is either possible to fix the normalization value (option **fix meanvalue** activated) or to automatically determine this value from the data (option **fix meanvalue** deactivated).

With the option **fix meanvalue** deactivated the program first calculates the mean amplitude within this window for all traces (this mean amplitude is also shown within the third filter parameter - not to change).

With the option **fix meanvalue** activated you must manually enter the mean amplitude value within the third filter parameter. This is useful if you want to compare different datafiles (e.g. using timeslices) which have different amplitude characteristics due to the source coupling.

The switch groupbox allows you to specify whether the amplitude equality distribution is performed within the **whole profile** or whether individual data parts are automatically determined for **line parts** separated by each other by a new ensemble number. In this case the scaling will be done for each ensemble independently.

Each time sample of a trace is then multiplied by a scaling factor, which is determined by the mean amplitude within the y-window of each trace and the previously calculated or determined mean total amplitude. The result is a mean amplitude equality distribution within the y-window for all traces.

The option can be employed reasonably, e.g. to compensate for different source couplings.

1.11.2.9 trace normalize

The filter acts on each trace independently. The gain filter facilitates the amplitude equality distribution in x-direction based on the max. amplitude of each trace.

Each time sample of a trace is then multiplied by a single scaling factor, which is determined by the total mean amplitude of the profile and the max. amplitude value of the trace. The result is a mean amplitude equality distribution for all traces.

With the option **restrict to +-1** activated the amplitude range will be rescaled to a range between -1 and +1. The output format will be set automatically to 32 bit floating point if 16 bit integer format is given as input format.

The option can be employed reasonably, e.g. to compensate for different source and receiver couplings.

1.11.2.10 x-distance decay(db)

The filter acts on each trace independently. The filter facilitates the compensation of energy losses in dependence from the shot-receiver distance x . All samples of one trace are scaled with the same filter value.

The filter function is linear with the distance x . The parameter **decay** defines the filter strength. The **0-distance scale** parameter defines a minimum scaling value in order to prevent that the scaling value becomes 0 for a shot-receiver distance of 0.

The option can be employed reasonably e.g. to compensate for energy losses because of different shot-receiver offsets (e.g. seismic shot data).

1.11.2.11 compensate stripes

This processing step is mainly designed for 3D-GPR data but it can also be applied on other data if the preconditions hold true. The step allows to compensate stripes which occur within timeslices if different antenna with different characteristics have been used for the data acquisition. A 2 and a 4 antenna configuration are supported. The data can have been acquired in normal and meandering mode. The different patterns must be chosen within the **pattern** group box. **12121212...** means that a 2 antenna configuration has been used with the same profile direction start coordinate of the 2D-lines. **12211221...**, **22112211...** and **21122112...** define the different 2 antenna configurations for the data acquired in meandering mode. The correct pattern for the meandering mode also depends on the overall timeslice start coordinate perpendicular to the profile direction.

Equivalent patterns are supported for a 4 antenna configuration but only one pattern for the meandering mode is given. Therefore the overall timeslice start coordinate perpendicular to the profile direction must correspond the real one or must be a multiple of $8 \cdot \text{scanincrement}$.

The **direction** box defines the direction of the stripes which is equivalent to the direction of the original 2D-lines of the timeslices.

The timeslices must have been created without interpolation inbetween the original 2D-lines. Therefore the original scanincrement must have been kept.

The **scaling factor ant.2** defines the factor for the lines acquired with antenna 2, the **scaling factor ant.3** defines the parameter for antenna 3 and **scaling factor ant.4** the factor for antenna 4. The amplitudes for antenna 1 will not be affected.

The option **apply on original data** allows you to control interactively the effect of the different scaling parameters within the original line (the data of the original line will not be changed).

The option can be employed reasonably e.g. to compensate for distinct stripe patterns within the timeslices if a multichannel GPR apparatus has been used.

1.11.2.12 normalize profiles

This processing step is designed for an energy equalization of different 2D-profiles. The processing step is only useful within the sequence processing for several profiles. If used for a single profile within the gain-menu no effect is given. The option is useful to compensate different coupling conditions, e.g. if an antenna array has been used for acquiring parallel 2D-lines.

Two different normalization methods can be chosen:

constant normalizing factor: one single scaling factor for each profile will be calculated from the quotient of the overall mean energy value of all profiles and the total energy value of the individual profile. The mean energy values will be determined within the given timewindow (options **start window** and **end window**). The option does not change the energy distribution within one single profile but only equalizes the total energy distribution between the chosen profiles.

Deactivating the option **all traces** allows you to restrict the tracerange for determining the normalizing factor. Only those traces which are defined by **1. trace** and **last trace** are taken into account to calculate the overall mean energy value of all profiles and the total energy value of the individual profile.

time-dependent normalizing curve: a timedependent scaling curve will be calculated for each profile depending on the overall mean envelope curve of all profiles and the mean envelope curve of the individual profile. The envelope curves will be determined over the total timerange and summed up over all traces (**option all traces active**) or over a given range of traces (**option 1. and last trace** with **option all traces deactivated**). Then this mean envelope (energy) curve will be smoothed over the given timerange (**option smooth range**). The option changes the energy distribution with respect to the timeaxis. For example: if a high reflection energy area exists within a special timerange for only few profiles the processing step will lower the energy within this timerange for these profiles.

To be considered: the option "time-dependent normalizing curve" may lead to a amplitude decrease of reflections which are only present in some profiles (e.g. pipes which are orientated parallel to the acquired profiles).

The option normalize profiles might be reasonably employed if e.g. calculated timeslices show distinct stripe patterns which indicate different coupling conditions. In contrast to the option compensate stripes (chap. 1.11.2.10) which acts on the timeslices itself the filter normalize profiles acts on the profile data and not on the timeslices. The new timeslices or the 3D-datafile must be generated afterwards.

1.11.2.13 normalize 3D-file

This processing step is designed for an energy equalization of a 3D-file. The option is useful to compensate different coupling conditions, e.g. if an antenna array has been used for acquiring the 3D-datafile.

Two different normalization methods can be chosen:

constant normalizing factor: one single scaling factor will be calculated for each 2D-ensemble of the 3D-file from the quotient of the overall mean energy value of the 3D-datafile and the total energy value of the individual 2D-ensemble. The option corresponds to the option `manual gain(x)` with suboption `get factors` for 3D-file (chap. 1.11.2.7). The mean energy values will be determined within the given timewindow (options **start window** and **end window**). The option does not change the energy distribution within one single ensemble but only equalizes the total energy distribution between the chosen profiles.

time-dependent normalizing curve: a timedependent scaling curve will be calculated for each 2D-ensemble of the 3D-file depending on the overall mean envelope curve of the complete 3D-file and the mean envelope curve of the individual 2D-ensemble. The envelope curves will be determined over the total timerange and summed up over all traces. Then this mean envelope (energy) curve will be smoothed over the given timerange (option **smooth range**). The option changes the energy distribution with respect to the timeaxis. For example: if a high reflection energy area exists within a special timerange for only few profiles the processing step will lower the energy within this timerange for these profiles.

To be considered: the option “time-dependent normalizing curve” may lead to a amplitude decrease of reflections which are only present in some profiles (e.g. pipes which are orientated parallel to the acquired profiles).

The option `normalize 3D-file` might be reasonably employed if e.g. the timeslices show distinct stripe patterns which indicate different coupling conditions.

1.11.3 StaticCorrection/muting

The following correction, muting and filter possibilities are included within the static correction/muting group:

- static correction
- dynamic correction
- move starttime
- muting
- surgical muting
- time cut
- correct max. phase
- corr.max.phase/wrap
- correct picked phase
- correct for 2 layers
- correct 3Dtopography

For some filters available within this group the original trace is shown together with the currently filtered trace, the spectrum of the original trace and the spectrum of the currently filtered trace. The arrows near the original trace and original spectrum are used to enhance the resolution of the trace and spectrum windows. The trace or the spectrum is stretched up and down when using these arrows. The spectrum scale is Hz, if the time scale is in milli-seconds and MHz, if the time scale is in nano-seconds, respectively.

In addition to the zooming arrows there exist an equivalent spin button which allows to move the starting point of the primary and filtered traces as well as of the corresponding spectra.

When activating the options static correction and muting a table appears. With all these options the inputs may be entered either interactively within the original profile or using the table input. The options save and load allow to store the current edit values to an ASCII file and to load the wanted edit values from file, respectively.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number**.

The filter parameters for the individual processing steps are entered within the **filter parameter group**. The up/down arrows near the input fields allow to change the actual entered value by + or - 10 %. With the option **Apply on example trace** activated the effect of the filter parameters is online controlled when changing the parameters by showing both the original and filtered trace. The arrows near the input fields may be best used for such an interactive control. Deactivate this parameter if this control takes too much computer time.

After having specified the filter parameter, you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing.

Activating the option **sequence proc.** additionally open the Sequence Processing menu. With this menu opened you may easily add the current

processing step to the sequence processing flow which may be applied to a choosable number of lines after.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing. Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing.

1.11.3.1 Static correction

The filter acts on each trace independently. The processing step static correction facilitates a static, this means a time-independent, correction for each trace in time direction.

The options move to positive times, move to negative times and move to positive times/shift control the direction of the static correction.

With the selection of **move to positive times** the traces are shifted down (i.e. towards larger times) exactly by the specified correction values. This processing step may also be used for a topographic correction. Normally for that purpose the topographic values must be manually changed into two-way traveltimes.

With the selection of **move to negative times** the traces are shifted up (i.e. towards smaller times) exactly by the specified correction values. It is reasonable to employ this option for example in order to invert a possible time delay.

With the selection of **move to negative times/shift** the largest static correction value is taken as the base level. The trace with this correction value is not modified in time. All traces are shifted down (i.e. towards larger times) relative to this base level according to the difference of its correction values and the base level. The inserted time range is filled with zeros. The shape of the correction is the same as for move to negative times but no data are lost with this selection. The choice move to negative times/shift therefore allows for example the compensation for topography under the assumption of a homogeneous velocity distribution. The topography only has to be transformed into two-way travel times for this. The smallest static correction value zero then matches the highest point in topography, the largest static correction value the lowest point in topography. The shape of the topography then matches exactly the shape of the correction values on screen.

Activating the option **use automatic filename** allows you to automatically read in the static correction values from an ASCII-file (see also option save for the format of the ASCII-file) with the filename correct and the fileextension equal to the profilename: Example: profilename: line1_.00t - correction filename: correct.line1_. The correctionfile must have been stored under the projectdirectory. Activating the option might be useful if you want to use the static correction within the sequence processing (see also chap. 1.11.11). Then no single correction ASCII-filename is queried but the correction values are read in from the different ASCII-files with the automatic filenames described above.

Deactivating the option **update tracedelays** will not change the original delay times stored within the individual traceheaders. This might be useful in order to reconstruct the original data (e.g. static correction/move to positive times and afterwards static correction/move to negative times both based on the traceheader delaytimes).

If activated the traceheader delaytimes will be changed according to the chosen static correction values.

After having activated the option **static correction** a table with the parameters distance and time appears which allows the interactive input of the static correction values. The **distance** and the **time** must be entered in the corresponding dimensions. Between the individual marks the values are linearly interpolated, at the x-margins the specified value belonging to the smallest or

largest x-coordinate, respectively, is extrapolated. The values can also be defined within the original profile by pressing the left mouse button at the desired position. The chosen values are automatically transferred to the table input and vice versa to the profile image.

The option **save** stores the current edit values on an ASCII file under the project directory with the name correct and a freely choosable extension. Each line of the ASCII file consists of two columns: 1. column: distance in the given distance dimension, 2. column: time in the given time dimension.

The option **load** loads the wanted edit values from an ASCII file with the filename correct. The table and profile image are automatically updated.

The option **reset** resets the current edit values.

The option **load pick data** allows to load new picks from disc. After having chosen a pick data set from the file list the picks are automatically converted into the table edit values. If no pick data set will be chosen the current picks are used and converted into the table edit values. Please note that all current edit values are lost in both cases.

The option **load trace delays** allows to read the traceheader time delay values into the static correction tabella. This option is useful in combination with the suboption move to positive times. If very large time delay values are present an overall fileheader starttime might be necessary.

Use this option for example for eliminating a time delay or trigger errors.

1.11.3.2 Dynamic correction

The filter acts on each trace independently. The processing step dynamic correction facilitates a dynamic, i.e. time dependent correction for a predefined constant velocity and the distance between source and receiver in time (y) direction.

The filter parameter **S/R-distance** determines the distance between source and receiver in the given distance dimension. The parameter **v** determines the constant velocity in the given velocity dimension. The calculated correction in time direction depends on the two entered parameters and the current time - the larger the time the smaller the correction.

If source and receiver have a large distance while recording the zero-offset profile the application of this option is recommended especially for near surface onsets.

1.11.3.3 Move starttime

The filter acts on each trace independently. The processing step move starttime facilitates a static correction in time direction by a given value (by default the **start time** specified within the fileheader time specification).

Within the groupbox move starttime you may define if the moving value is always taken from the fileheader (option **fileheader** activated) or whether a manual input is possible (**manual input** activated).

The filter parameter **move time** determines the time value each trace is shifted in the case the option manual input is activated. The default value is the start time value stored in the fileheader.

After having performed the processing step the start time of the processed file is automatically set to 0.

When using this option within the Sequence Processing with activated option fileheader, always the start time values stored in the individual fileheaders are used. With activated option manual input the entered constant value is used for all files.

You may use this option for removing all data in time before the start time or you may use this option as a simple static correction (see also static correction).

1.11.3.4 Muting

The filter acts on each trace independently. The processing step static correction facilitates the possibility of defining a muting curve above or below which all data points are set to zero.

With the selection of **mute data above muting curve** all data points above the specified muting curve (towards smaller times) are set to 0.

With the selection of **mute data below muting curve** all data points below the specified muting curve towards larger times) are set to 0.

The filter parameter **muting window** in the given time dimension defines the timerange for the muting. Only these datapoints which fall inside this timerange below or above the given muting curve are muted. By default the muting window is set to the complete timerange of the profile.

The filter parameter **taper window** in the given time dimension determines a time window for a smooth muting. The data within this timewindow above or below the muting curve (dependent from the settings above) are tapered using a cos-function. A value of 0 means that the data are truncated at the muting curve.

After having activated the option **muting** a table with the parameters distance and time appears which allows the interactive input of the new muting curve. The **distance** and the **time** must be entered in the corresponding dimensions. Between the individual marks the values are linearly interpolated, at the x-margins the specified value belonging to the smallest or largest x-coordinate, respectively, is extrapolated. The values can also be defined within the original profile by pressing the left mouse button at the desired position. The chosen values are automatically transferred to the table input and vice versa to the profile image.

The option **apply on whole line** allows the current chosen muting points (starting at trace 1) to be periodically repeated for the complete line. Using this option is reasonable for defining a muting for a shot gather in the case all shots all combined together within one file. You only have to define the muting curve for the first single ensemble, e.g. for the first shot and then use this option to apply the current muting curve on all existing shots. To be considered: the geometry for all shots must be the same.

The option **save** stores the current edit values on an ASCII file under the project directory with the name correct and a freely choosable extension. Each line of the ASCII file consists of two columns: 1. column: distance in the given distance dimension, 2. column: time in the given time dimension.

The option **load** loads the wanted edit values from an ASCII file with the filename correct. The table and profile image are automatically updated.

The option **reset** resets the current edit values.

The option **load pick data** allows to load new picks from disc. After having chosen a pick data set from the file list the picks are automatically converted into the table edit values. Please note that all current edit values are lost.

1.11.3.5 surgical muting

The filter acts on each trace independently. The processing step surgical muting facilitates the muting below or above trajectories of a given velocity.

The following possibilities are available:

- 1: **Inside mute** - edit data between trajectories defined by 1.velocity and 2.velocity. The data inside the fan will be muted.
- 2: **Top mute** - edit data above mute trajectory defined by 1.velocity.
- 3: **Bottom mute** - edit data below mute trajectory defined by 1.velocity.
- 4: **Around mute** - edit data around trajectories defined by 1.velocity and 2.velocity.
- 5: **Outside mute** - edit data between trajectories defined by 1.velocity and 2.velocity. The data outside the fan will be muted.

The trajectories are calculated based on the entered velocities and the shot and the receiver traceheader coordinates.

The filter parameter **muting window** in the given time dimension defines the timerange for the muting. The option is enabled for top, bottom and around mute.

The filter parameter **time shift** in the given time dimension defines a time shift for the lower velocity muting curve. The option is enabled for inside and outside. A useful value is for example the signal length or a multiple of it.

The filter parameter **taper window** in the given time dimension determines a time window for a smooth muting. The data within this timewindow above or below the muting curve (dependent from the settings above) are tapered using a cos-function. A value of 0 means that the data are truncated at the muting curve.

1.11.3.6 Time cut

The filter acts on each trace independently. The processing step time cut offers the possibility to limit or to extend each trace to a predefinable maximum time.

The filter parameter **max.time** in the given time dimension determines the time each trace is limited to. No compression or resorting takes place. Only the time range is reduced to a preset range.

The filter parameter **fill for ext. times (0,1)** controls how the filling is done if the max. time is larger than the original measured time. Enter 0 for filling with zero values and enter 1 for filling with the last original value (useful e.g. for rastered models).

Use this option for example if the time range was set to a too large range during the data acquirement and you want to save disk space and enhance the computer speed for the further data processing.

1.11.3.7 Correct max. phase

The filter acts on each trace independently. The processing step **correct max. phase** offers the possibility to automatically find the phase of max. amplitude within a given time window and to flatten this phase. An automatic resampling in time direction for each trace is included during the processing in order to increase the resolution of the automatically determined phase.

The filter parameters **start window** and **end window** in the given time dimension determine the time window which is used for finding the phase to be corrected for. The **polarity** determines the polarity of the phase. The program determines for each trace the maximum amplitude having the chosen polarity within the given timewindow.

With the option **first onset** under **polarity** activated the automatically determined first arrival (independent from the polarity) is used for the correction. The input parameter is a value **threshold** (absolute value) and the dominant **frequency** of the wavelet. If the polarity of the automatically determined onsets has been changed for one or more traces a warning message appears.

With the option **zero cross.** activated the identified time is shifted to smaller times to the zero crossing.

The static correction is done for the corresponding time value for each trace. Analogous to the suboption move to negative times/shift of the static correction the traces are not shifted by this value but a **mean time** is determined first which is taken as the base level.

The parameter **average(traces)** is used for building up a rolling median of the time shifts for each trace. Enter a value greater than 2 if there are some traces where the max.phase is disturbed within the given time window and you want to use the median time shift built up from the neighbouring traces.

The switch group **correct time** is used for setting the time base. It controls the shifting of the start time within the file header. With correct time set to auto the time base is set to 0 for the flattened phase. This is done by automatically setting the start time (see also fileheader time specification) to the negative value of the mean time. With correct time set to no the start time is not changed. With correct time set to manual you must enter a **target time** within the filter parameter group. The time base of the flattened phase is set to that target time.

With the switch group **use mean** you may specify if only the mean time shift is used for all traces (set to yes) or if each trace is moved by its individual time shift (set to no) or if the mean time shift is determined for lineparts separated by each other by a new ensemble number (e.g. a 3D-datafile). With use mean set to yes the mean time shift is calculated and only the start time is changed depending on the options set within the switch group correct time. To be considered: if correct time is set to no nothing happens for this particular case.

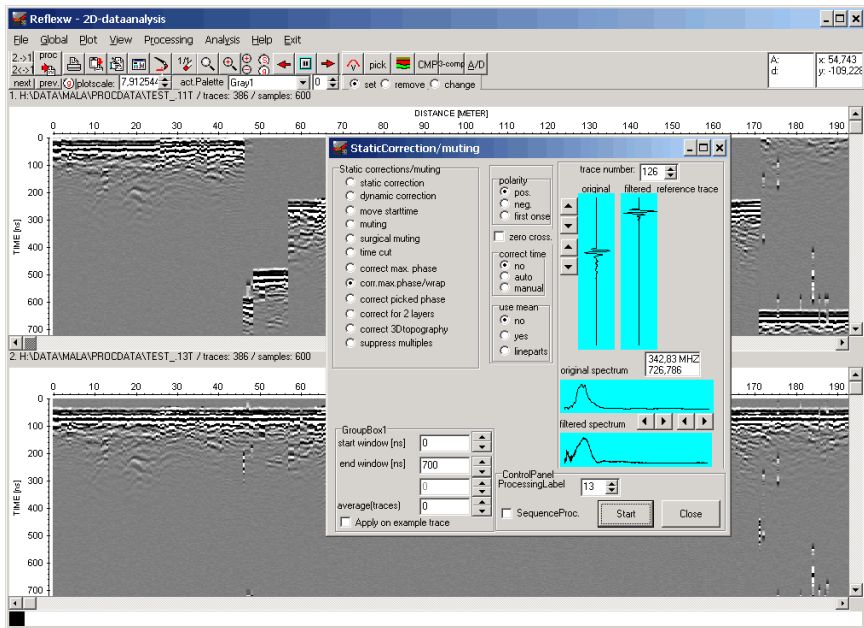
Use this option for example for a fast static correction of the first arrival (e.g. the direct wave from source to the receiver) if this phase shows some irregularities (e.g. a drift or trigger fluctuations) or to automatically change the start time.

1.11.3.8 Corr.max. phase/wrap

The filter acts on each trace independently. The filter performs a static correction corresponding to correct max.phase described under chap. 1.11.3.7. In addition the data above the detected correction phase which are normally omitted are wrapped at the end (timeend) of the data. The timeshift is only set to 1/4 of the total detected timeshift in order to keep most of the data.

One application of the filter might be a compensation of trigger problems (see picture below). For that purpose the timewindow for detecting the max. phase must be great enough and of course the max. phase must be consistent. Therefore in most case only the first arrival without any gain applies for this precondition.

The filter parameters are the same like for the option correct max. phase.



1.11.3.9 Correct picked phase

The filter acts on each trace independently. The processing step performs a static correction (see also static correction) of a picked phase stored under the same name as the data filename.

Within the group box **correct phase** you may determine how the picked phase is changed.

With the option **flatten** activated the picked phase is flattened. For example this is useful for flattening the first arrival because of a continuous timeshift of the start pulse.

With the option **lowpass** activated you are able to lowpass filter the picked phase. For that purpose you must enter the upper cutoff wavelength (option **wavelength**) in the given distance dimension. Fluctuations in x-direction, which are smaller than this wavelength, will be eliminated. The option may be used for the smoothing of small scale topography effects or for the smoothing of the swell. Extensive fluctuations will be kept.

With the option **highpass** activated you are able to highpass filter the picked phase. For that purpose you must enter the lower cutoff wavelength (option **wavelength**) in the given distance dimension. Fluctuations in x-direction, which are larger than this wavelength, will be eliminated. The option may be used for the smoothing of large scale topography effects.

With the option **move pos.** activated the traces are shifted down (i.e. towards larger times) exactly by the specified correction pickvalues. This processing step may also be used for a topographic correction. Normally for that purpose the picks must be manually changed into two-way traveltimes.

With the option **flatten** or **highpass** activated you must set the **target time** within the filter parameter group. The time base of the flattened or filtered phase is set to that target time. The start time (see also file header time specification) is automatically changed based on the difference between the entered target time and the largest pick time value.

The option **wavelength** determines the upper or lower cutoff wavelength if the options **lowpass** or **highpass** are activated.

1.11.3.10 Correct for 2 layers

The filter acts on each trace independently. The processing step static correct for 2 layers facilitates a static, this means a time-independent correction for each trace in time direction based on a given 2 layer model.

The processing step is used for a correction of a rough surface if the data are acquired in a flat niveau above this surface. The idea of this static correction step is that the area between the data acquisition niveau and the surface shall be replaced by the material with the same velocity as below the surface. Therefore the velocities within the layers above and below the surface must be known (see filter parameter 1. velocity and 2. velocity) and the correction values must be entered for the surface reflection.

The filter parameter **min time** in the given time dimension determines the time base for the static shift. This time should be smaller than the minimum time value of the correction timevalues (see below) and must be equal for all lines which are interpreted together within a subsequent step (e.g. calculation of timeslices). Analogous to the suboption move to negative times/shift of the static correction the traces are not shifted by the individual correction timevalues but this min. time is taken as the base level.

The filter parameter **1. velocity** determines the velocity within the layer above the correction timevalues.

The filter parameter **2. velocity** determines the velocity within the layer below the correction timevalues.

After having activated the option **correct for 2 layers** a table with the parameters distance and time appears which allows the interactive input of the static correction values. The **distance** and the **time** must be entered in the corresponding dimensions. Between the individual marks the values are linearly interpolated, at the x-margins the specified value belonging to the smallest or largest x-coordinate, respectively, is extrapolated. The values can also be defined within the original profile by pressing the left mouse button at the desired position. The chosen values are automatically transferred to the table input and vice versa to the profile image.

The option **save** stores the current edit values on an ASCII file under the project directory with the name correct and a freely choosable extension. Each line of the ASCII file consists of two columns: 1. column: distance in the given distance dimension, 2. column: time in the given time dimension.

The option **load** loads the wanted edit values from an ASCII file with the filename correct. The table and profile image are automatically updated.

The option **reset** resets the current edit values.

The option **load pick data** allows to load new picks from disc. After having chosen a pick data set from the file list the picks are automatically converted into the table edit values. Please note that all current edit values are lost.

Use this option for example for remove the influence of a rough surface if the data have been acquired in a flat niveau above the rough surface.

1.11.3.11 Correct 3D-topography

The processing step correct 3D-topography facilitates a static, this means a time-independent, correction for each trace in time direction based on 3D topography coordinates stored within an ASCII-file or based on the z-traceheader coordinates.

With the option **z-tracecoord** activated the static correction is based on the shot and the receiver elevations stored within the traceheader of each trace (see also `plotoption correct header elevations`).

The filter parameter **velocity** is used to transform the z-values of the topographic coordinates into a two-way travel time whereby the traveltime shift value is calculated from the sum of the shot and the receiver elevation differences. If the original section represents a depth section (e.g. after `timedepth` conversion) the velocity should be set to a value of 2 because in this case no transformation into two-way traveltimes are necessary. Using a value of 2 results in no rescaling. If the y-dimension is identical to the x-dimension (normally the case after a time depth conversion of for a depth section) the velocity will be automatically set to 2 and the parameters velocity is disabled.

With the option **ASCII-file** activated the correction is based on an ASCII-file. Two ASCII-file formats are supported.

3D-topography format (filename `top*.*`):

Each line of the ASCII-file with the 3D topographic coordinates contains one set of 3D coordinates:

x-position y-position z-position

The x- and y-position define the acquisition plane where the profiles are located.

The z-position defines the topographic niveau. At least one blank must exist between the individual coordinates. Example with 3 different coordinate sets:

```
0 0 10
1 0 9
2 0 9.5
```

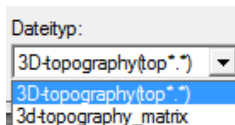
The ASCII-file may also consist of only the two coordinates x and z. In this case the y-coordinate will be automatically set to 0.

3d-topography matrix (filename `matrix_top*.*`):

The format is an ASCII matrix format which contains all values in x-direction within one line. The increment between the individual values is fixed and the same for both directions. There are 5 header rows. The first specifies the number of columns, the second the number rows. The third defines the right (min.) x-coordinate, the fourth the lowermost (max) y-coordinate. The fifth column contains the increment which is negative for the y-coordinates.

Example:

```
ncols      3312
nrows      4269
xlcorner   -303320.000000000000
ylcorner   1972570.000000000000
cellsize    2.000000000000
148.796875 148.78125 148.7578125 148.7421875 148.7109375 148.6875
148.65625 .....
148.6171875 148.578125 148.5703125 148.5625 148.546875 148.53125
148.515625 148.5 .....
```



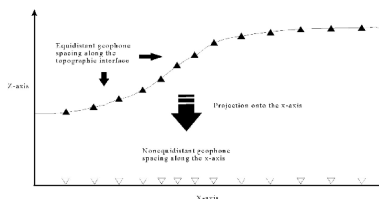
The dimension of the coordinates must be the same like within the datafile(s). To be considered: the profiles must be located within the x-y plane, this means the profile direction and the profile constant must be x and y or vice versa. The option **load** allows to specify the wanted topography file. The filename should begin with "top" and the file should be stored under the projectdirectory. The filter parameters **x-reference** and **y-reference** define a reference point which is subtracted from all topographic coordinates. Therefore it is not necessary to input the topographic coordinates using the same coordinate level like for the profile coordinates.

The filter parameter **velocity** is used to transform the z-values of the topographic coordinates into a two-way travel time. If the original section represents a depth section (e.g. after timedepth conversion) the velocity will be automatically set to 2 and the parameters velocity is disabled. In this case no transformation into two-way traveltimes are necessary. Using a value of 2 results in no rescaling.

The coordinates are either taken from the **fileheader** or from the **traceheader** (shot and receiver coordinates).

The spatial interpolation is either linear (**lin.interpol.** activated) or quadratic (**sqr.interpol.** activated).

The program automatically determines the individual topographic values from the given topography matrix. For this purpose the next topographic coordinate set within each quadrant is determined for each trace location. Afterwards a weighted mean of the single z-positions is determined which is used for the static correction for this trace.



The box correction allows to specify whether the topographic values are given in **altitudes** (default) or **depths**.

The correction corresponds to static correction-move to negative times/shift, i.e. the greatest z-

position (transformed into two-way travel time) is taken as the base level. This greatest z-position is written out within the processing flow (3.parameter z-reference). The trace with this correction value is not modified in time. With the option **manual z-reference** activated this base level must be entered manually within the parameter z-reference. This allows you to define the same base level for different profiles, e.g. if a 3D-dataset shall be constructed later.

All traces are shifted down (i.e. towards larger times) relative to this base level according to the difference of its correction values and the base level. The inserted time range is filled with zeros. This means that the area between the current topographic niveau and the maximum niveau is filled up with material with the given velocity.

With the option **project on x- or y-axis** activated the geometries of the shots and the receivers are recalculated based on the topographic xyz-values for profiles which are strictly orientated into the x- or y-direction. It is assumed that the original x- or y-coordinates do not represent the correct x- or y-coordinates but

had been determined directly on the topographic interface. The program automatically determines the positions of all shots and receivers on the given topography and calculates the x- or y-(for profiles which are orientated into the y-direction) projections of these positions respectively. The x- and y-coordinates of the topographically corrected data represent the true x- and y-coordinate within the xyz-coordinate system. The projection normally leads to a reduction of the total length of the profile.

To be considered: the x- and y-coordinates within the loaded ASCII-file must also represent true x- and y-coordinates within the xyz-coordinate system. They also must have been adapted to the original profile coordinates.

A following example tries to explain how the start position of the projection is defined and based on this how to set the ASCII-file. Three cases will be distinguished:

1. All coordinates are positive: the start coordinate of the profile remains unchanged and therefore must represent the projected x- or y-coordinate.
2. All coordinates are negative: the end coordinate of the profile remains unchanged and therefore must represent the projected x- or y-coordinate.
3. The coordinate range from negative to positive: the 0 position remains unchanged and therefore must represent the projected x- or y-coordinate.

Example:

The profilelength along the topographic interface is 10m. The elevation ranges from 1 to 6 m, this means the slope is 30° .

The profile for the 1 example ranges from 10-20 m, for the 2. example from -20 to -10 m and for the 3. example from -5 to 5 m.

The resulting projection length is in all cases 8.66 m ($10 \text{ m} * \cos(30^\circ)$)

start and end ≥ 0

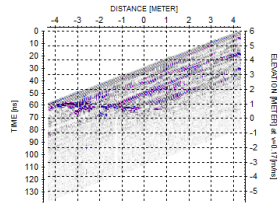
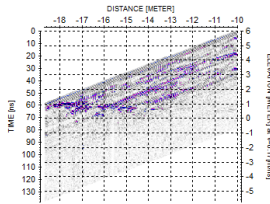
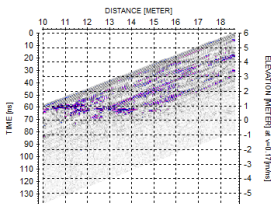
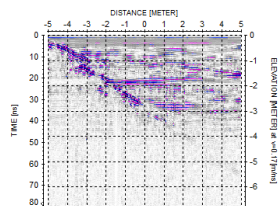
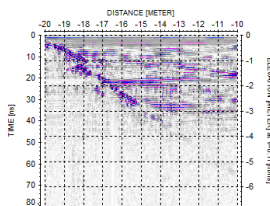
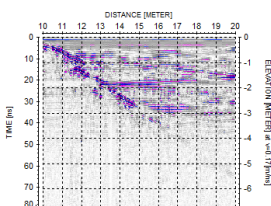
10.00 0 1.0
18.66 0 6.0

start and end ≤ 0

-18.66 0 1.0
-10.00 0 6.0

start ≤ 0 and end ≥ 0

-4.33 0 1.0
4.33 0 6.0



The mean trace increment will be updated. The endcoordinate will be updated if all coordinates are positive. If all coordinates are negative the start coordinate will be updated and for negative to positive coordinates end and start coordinates will be updated (0 will remain at the same position). The traceheader coordinates as well as the traceheader distances contain the true coordinates. With the suboption **make equidistant** equidistant traces will be created based on the traceheader distances. The mean traceincrement will be used for the construction. The use of this option is not recommended for data with a quite large increment and changing source/receiver increments like e.g. seismic refraction data as the necessary interpolation for creating equidistant data will be done for identical time values (horizontal interpolation).

With the option **only project - no correct** activated no topographic correction will be done but only a projection either on the x- or y-axis (depending on the profile direction). Use this option for example if you want to perform a topographic migration afterwards and your surface topographic values are given as pure xyz values (not recorded on the surface topography).

With the option **update z-coord.only** activated no topographic correction will be done but only the z-traceheader coordinates are updated. The option is only useful if the coordinates are read from an ASCII-file.

Pecularity for activated options **z-tracecoord.** and **traceheader::** the source and receiver coordinates are not changed using the option **project on x- or y-axis** but the distances are only recalculated based on these coordinates.

The option is also available within the sequence processing.

1.11.3.12 Suppress multiples

The option allows to suppress multiple reflections from the surface. For that purpose each wanted multiple reflection is first flattened based on the picked primary reflection and then the subtracting average filter is applied within a choosable time range around the flattened multiple.

The picks of the primary reflection must be saved under the same filename as the current profile filename (option use filename automatic name within the save pick menu activated).

The following inputs are necessary:

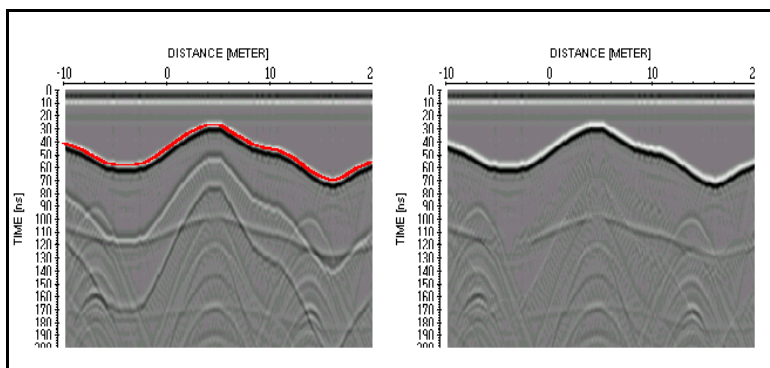
time range: specifies the time range for applying the subtracting average around the flattened multiple reflection. The range should be greater or equal the signal length.

time lag: enter a lag for shifting the time range to smaller times. Such a lag is necessary if you did not pick the very first arrival of the primary reflection. A default value is about a quarter of the signal length if the first maximum has been picked.

nr. of multiples: enter the number of multiple reflections to be suppressed.

nr. of traces: enter the number of traces for the subtracting average filter. The smaller this value the larger is the effect of the subtracting average filter (see also processing step subtracting average).

If the option **primary phase** within the suppress panel is activated the primary



phase will be suppressed in addition. The option might be useful for the elimination of the air-surface reflection from e.g. air coupled horn antenna.

The picture shows an example of the suppress multiples filter. The left panel shows the original data together with the picks of the primary reflection, the right panel shows the filtered data (time range: 20 ns, time lag: 10 ns, nr. of multiples: 3, nr. of traces: 20).

The option is not available within the sequence processing. The option is available for the **batch start** processing.

1.11.4 Declipping/arithmetic functions

The following filter possibilities are included in the declipping/arithmetic functions group:

declipping:

declipping/max.value

declipping/plateau

declipping/threshold

unwrapping

arithmetic function:

addition

multiplication

square

root

natural log

absolute

exp x

reciprocal (1/value)

set ampl.range to 0

conv. v to epsilon

For all filters available within this group the original trace is shown together with the currently filtered trace, the spectrum of the original trace and the spectrum of the currently filtered trace.

The arrows near the original trace and original spectrum are used to enhance the resolution of the trace and spectrum windows. The trace or the spectrum is stretched up and down when using these arrows. The spectrum scale is Hz, if the time scale is in milli-seconds and MHz, if the time scale is in nano-seconds, respectively.

In addition to the zooming arrows there exist an equivalent spin button which allows to move the starting point of the primary and filtered traces as well as of the corresponding spectra.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number** or by clicking in the primary file using the left mouse button.

The filter parameters for the individual processing steps are entered within the **filter parameter group**. The up/down arrows near the input fields allow to change the actual entered value by + or - 10 %. With the option **Apply on example trace** activated the effect of the filter parameters is online controlled when changing the parameters by showing both the original and filtered trace. The arrows near the input fields may be best used for such an interactive control. Deactivate this parameter if this control takes too much computer time.

After having specified the filter parameter, you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing.

Activating the option **sequence proc.** additionally open the Sequence Processing menu. With this menu opened you may easily add the current

processing step to the sequence processing flow which may be applied to a choosable number of lines after.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing. Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing.

1.11.4.1 Declipping

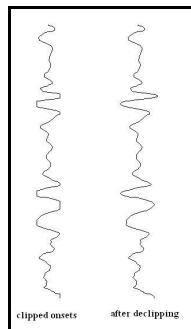
The filter acts on each trace independently. The filter enables a declipping of special onsets (see the different kinds of clipping). The declipping is done using a polynome of the 3. order based on the 4 next neighbours.

Depending on your data you may choose between different kinds of declipping in order to automatically specify which data are clipped and which data are not clipped:

Choosing **declipping/max.value** is useful if your clipped data always have the max. value existing in your data file (see figure on the right).

Declipping/plateau should be used if the clipped data have a plateau, this means that all amplitude values for that special signal part are equal (again see figure on the right).

Declipping/threshold should only be used if the clipped data are disturbed. In some cases the clipped data jump from positive to negative values. In these cases you should use the threshold declipping.



If choosing declipping/max.value and declipping/plateau declipping is done if at least two successive samples do have the same amplitudes. Choosing the suboption max.value the amplitudes values additionally must be equal to the maximum value given in the complete profile. Choosing the suboption plateau no additional restriction concerning the amplitude value is given.

Choosing the suboption threshold, a **threshold** value must be entered. All amplitudes greater/equal than this value are clipped per definition. You may choose the suboption threshold if no unified plateau value is reached or if the amplitudes did change significantly during a previous processing step (for example application of a gain-function or an AGC).

After the declipping all amplitude values are automatically multiplied by a **scaling factor**. You should choose this factor in such a way that all amplitude values lie within the 16 data bits if your REFLEX format is 16 bit Integer. If not, those amplitudes are automatically restricted to values of 32000 or -32000 in this case. If your REFLEX format is 32 bit floating point, the scaling value can be set to 1.

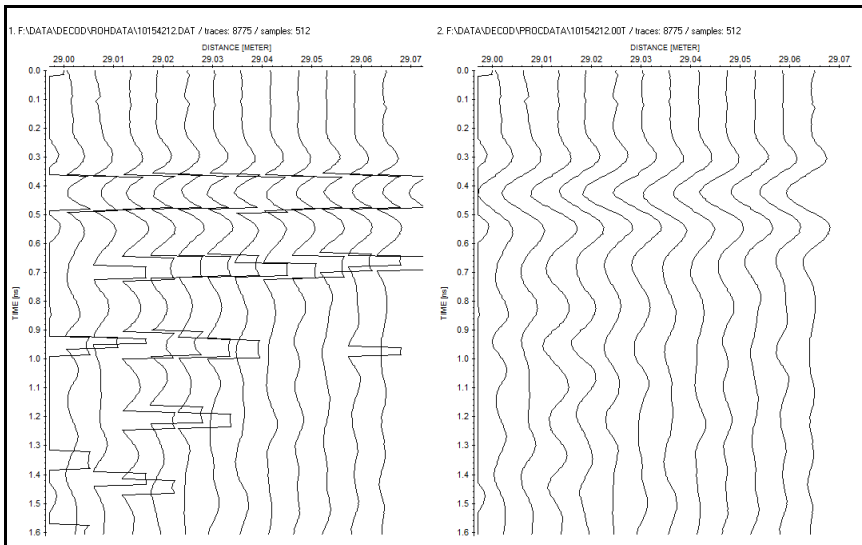
1.11.4.2 Unwrapping

The filter acts on each trace independently. The filter enables an unwrapping for all values showing a wrapping due to an insufficient data representation

The **wrapping value** (positive) value for which the wrapping occurs is by default 32768 assuming original 16 bit integer data.

You may choose the suboption threshold if no unified plateau value is reached or if the amplitudes did change significantly

After the unwrapping all amplitude values are automatically multiplied by a **scaling factor**. You should choose this factor in such a way that all amplitude values lie within the 16 data bits if your REFLEX format is 16 bit Integer. If not those amplitudes are automatically restricted to values of 32000 or -32000 in this case. If your REFLEX format is 32 bit floating point the scaling value can be set to 1.



1.11.4.3 Arithmetic functions

The filter acts on each trace independently. Under this processing step you can apply simple mathematical operations always acting only on the current sample of a trace.

The following arithmetic functions are available:
addition, multiplication, square, root, natural log, absolute and exp x.

The filter **addition** requires the input of a value which is added to every data point. Doing this, e.g. the representation of exclusively negative data can be improved through the addition of a positive value, if the data after the addition fluctuate around zero.

The filter **multiplication** requires the input of a value each data point is multiplied with. This can be employed reasonably if the amplitude level has become too large or too small after a data processing step. Another possible application is the amplitude normalization of different data sets. Within the **distance range** panel you may restrict the traces for which the filter will be applied. With all traces activated all traces within the profile will be taken into account. With deactivated option you may enter the 1. and the last trace for applying the filter. The filter parameters **start time and end time** allow to restrict the range in time (y)-direction.

The filters **square**, **root** and **natural log** can be used to emphasize certain amplitude ranges. The sign of each data point is maintained when applying this option. So, the option square for example improves the resolution of the absolute largest amplitudes.

With the filter **absolute** each data point receives a positive sign.

The filter **exp x** exponentiates all data values by x. You have to specify the exponent x as a filter parameter.

The filter **reciprocal (1/value)** generates the reciprocal of each value. The filter might be used for example to create slowness from velocity values and vice versa.

The filter **set ampl.range to 0** allows to set to 0 all data values lying inbetween the given amplitude range. You have to specify the amplitude range by the values min. amplitude and max. amplitude. Example: if you want to set the complete negative phase to 0, enter for min. amplitude the highest negative value existing in the profile and for max. amplitude 0.

The filter **conv. v to epsilon** converts velocities with m/ns into epsilon values and vice versa using the following formula: $v=0.3/\sqrt{\epsilon}$. With the Filterparameter set to 0 the conversion is done for v[m/ns] to epsilon, if set to 1 the conversion is done for epsilon to v[m/ns].

1.11.5 Complex trace-analysis/spectral analysis

The following filter possibilities are included in the complex trace-analysis/spectral analysis functions group:

complex trace-analysis
trace spectrum
moving window spectrum
rotate 2 components
add random noise
trace phase-spectrum
texture attribute

For some filters available within this group the original trace is shown together with the currently filtered trace, the spectrum of the original trace and the spectrum of the currently filtered trace.

The arrows near the original trace and original spectrum are used to enhance the resolution of the trace and spectrum windows. The trace or the spectrum is stretched up and down when using these arrows. The spectrum scale is Hz, if the time scale is in milli-seconds and MHz, if the time scale is in nano-seconds, respectively.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number** or by clicking in the primary file using the left mouse button.

The filter parameters for the individual processing steps are entered within the **filter parameter group**. The up/down arrows near the input fields allow to change the actual entered value by + or - 10 %. With the option **Apply on example trace** activated the effect of the filter parameters is online controlled when changing the parameters by showing both the original and filtered trace. The arrows near the input fields may be best used for such an interactive control. Deactivate this parameter if this control takes too much computer time.

After having specified the filter parameter you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing.

Activating the option **sequence proc.** additionally open the Sequence Processing menu. With this menu opened you may easily add the current processing step to the sequence processing flow which may be applied to a choosable number of lines after.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing. Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing. Exceptions:
processing step **insert profile**: here an additional character(s) is asked which will be automatically added to the actual filenames for the creation of the secondary lines which shall be inserted. Example:

actual filename: data_gps.00t - additional characters: _insert -> secondary file:
data_gps_insert

1.11.5.1 Complex trace-analysis

The filter acts on each trace independently. The Hilbert-Transformation is used to calculate **instantaneous attributes** like the envelope, the instantaneous frequency or the instantaneous phase.

The instantaneous amplitude or **envelope** is a measure for the reflectivity strength, which is proportional to the square root of the complete energy of the signal at an instant of time. The envelope gives an overview of the energy distribution of the traces and on the other hand it can facilitate the determination of signal first arrivals.

The **instantaneous phase** is a measure of the continuity of events on a section. Jumps within the instantaneous phase might indicate superpositions of arrivals for example. The phase can be plotted continuously with the option phase/cont. or limited to an interval of 2π with the option phase/2pi. For the display a normalized wiggle-mode is advantageous. Because of the normally small values the 32 bit floating point format is automatically chosen for the processed data.

The temporal rate of change of the instantaneous phase is the **instantaneous frequency**. The instantaneous frequency is calculated as the temporal derivative of the phase using an approximation by central differences. The plot of the instantaneous frequency allows to recognize superpositions of arrivals, since different arrivals are often characterized by different instantaneous frequencies. For this the choice of a normalized wiggle-mode is often more useful as the point-mode. The instantaneous frequency may have a high degree of variation which may or may not be related to stratigraphy. Therefore, an **average** filter parameter is queried. It determines over how many samples the calculated instantaneous frequencies should be averaged. The calculation of a mean value is reasonable because of the often strong fluctuations of the instantaneous frequencies. Because of the normally small values the 32 bit floating point format is automatically chosen for the processed data.

1.11.5.2 Trace spectrum

The filter acts on each trace independently. The filter **trace-spectrum** allows the calculation of the amplitude spectrum of each trace.

It is possible to restrict the amplitude spectrum calculation to a distinct time range. For this you must enter within the filter parameter group the **start time** and the **end time** for which the calculation shall be done. By default start time is set to the time-begin of each trace and end time is set to the max. time of each trace.

The option **taperlength** (in the given timedimension) allows to enter a taper range at the beginning and at the end of the specified timerange.

The y-axis label of the filtered section automatically changes to frequency.

The option **max. frequency** allows to restrict the maximum frequency for the resulting trace spectrum. The default value depends on the original timeincrement.

The sampling rate of the resulting frequency axis depends on the total time range of the original profile. The option **frequency resampling** allows to enter a factor for the resampling of the resulting frequency axis. Enter a greater factor than 1 if the frequency axis is undersampled.

1.11.5.3 Moving window spectrum

The filter acts on a choosable number of traces. The filter **moving window spectrum** allows a moving-window frequency analysis for all time samples averaged over all traces or for each trace separately. Thus, you get a time-frequency dependency. The dependency of the distance (x-axis) is eliminated.

Here, for each time value the amplitude spectrum is calculated over a window to be specified (filter parameter **window width** in the corresponding time dimension). The window is moved continuously over the y-(time)axis whereby only every n. sample is used (filter parameter **every n. sample**). With the specification of a value greater than 1 for **every n. sample** you have the possibility to have the computation-time consuming analysis performed not for each time value but only for each n..time sample.

The parameters **start time (y)** and **end time (y)** define the time range for the moving window spectrum analysis.

Within the **distance range** panel you may restrict the traces for which the filter will be applied. With **all traces** activated all traces within the profile will be taken into account. With deactivated option you may enter the **1.** and the **last trace** for applying the processing step.

With the option **apply on each trace separately** deactivated the amplitude spectra are summed contemporaneously for the choosen number of traces.

With the option **apply on each trace separately** activated the moving spectrum for each trace is calculated separately without any summing. After having started the option the overall filename must be entered. An automatic labelling is added to the filename depending on the trace number of the original file. Example with 100 different independent files: entered new overall filename: test - the new filenames will be: test0001.00t, test0002.00t,.....,test0100.00t. The files are stored under the subdirectory procdata with the extension 00t.

The sampling rate of the resulting frequency axis depends on the entered window width. The option **frequency resampling** allows to enter a factor for the resampling of the resulting frequency axis. Enter a greater factor than 1 if the frequency axis is undersampled.

The moving-spectral analysis allows the visualization of frequency range shiftings with time (y-coordinate). This is reasonable for example for an attenuation analysis or for the recognition of a possibly frequency-dependent reflection or transmission behaviour of the investigation area.

1.11.5.4 Rotate 2 components

The filter **rotate 2 components** allows to rotate two components around an angle given within the traceheader under CMP(x-pos.).

The data must be 3-component dipole/frame data with the first component (1., 4., 7., ... trace) containing the data from the dipole antenna and the 2. (2., 5., 8., ... trace) and 3. component (3., 6., 9., ... trace) containing the data from two frame antenna. The data from the dipole antenna remain unchanged whereas the data from the two frame antenna are rotated around the angle given within the traceheader (CMP-x-pos.) - see also 3-component analysis (chap. 1.12.8).

1.11.5.5 Add random noise

The filter **add random noise** allows to add random noise. This might be useful e.g. for synthetic data in order to get a more realistic result. The noise is uncorrelated.

The filterparameter **noise percentage** controls the strength of the noise. The percentage input refers to the max. value within the original profile. Four different **noise types** are supported: white, normal, exponential and self similar. In each case the filter parameter **nominal frequency** controls the dominant frequency used for the noise type.

1.11.5.6 Trace phase-spectrum

The filter acts on each trace independently. The filter **trace phase-spectrum** allows the calculation of the phase spectrum of each trace.

It is possible to restrict the phase spectrum calculation to a distinct time range. For this you must enter within the filter parameter group the **start time** and the **end time** for which the calculation shall be done. By default start time is set to the time-begin of each trace and end time is set to the max. time of each trace.

The option **taperlength** (in the given timedimension) allows to enter a taperrange at the beginning and at the end of the specified timerange.

The y-axis label of the filtered section automatically changes to frequency. The phase is restricted to the range between 0 and 2 pi. Therefore a jump occurs when the phase changes from a high value around 2 pi to a small value.

The option **max. frequency** allows to restrict the maximum frequency for the resulting trace spectrum. The default value depends on the original timeincrement.

The sampling rate of the resulting frequency axis depends on the total time range of the original profile. The option **frequency resampling** allows to enter a factor for the resampling of the resulting frequency axis. Enter a greater factor than 1 if the frequency axis is undersampled.

1.11.5.7 texture attribute

The option extracts textural features using a co-occurrence matrix depicting spatial relations or patterns of neighboring amplitudes.

First the data are transformed into gray level whereby the total number of gray levels is given by **nr. gray levels**. Then a co-occurrence matrix is built for a texture element of given size (**size in traces** and **size in samples**) at each position (moving window). The size in samples should cover approximately one period and will be automatically used as default value if given within the fileheader. A useful value for the size in traces is around 10.

Four possible directions (**horizontal**, **vertical**, **45 degree** and **135 degree**) may be used for creating the co-occurrence matrix. Horizontal for example means that the neighboring grey levels at the right are summed up for the matrix building.

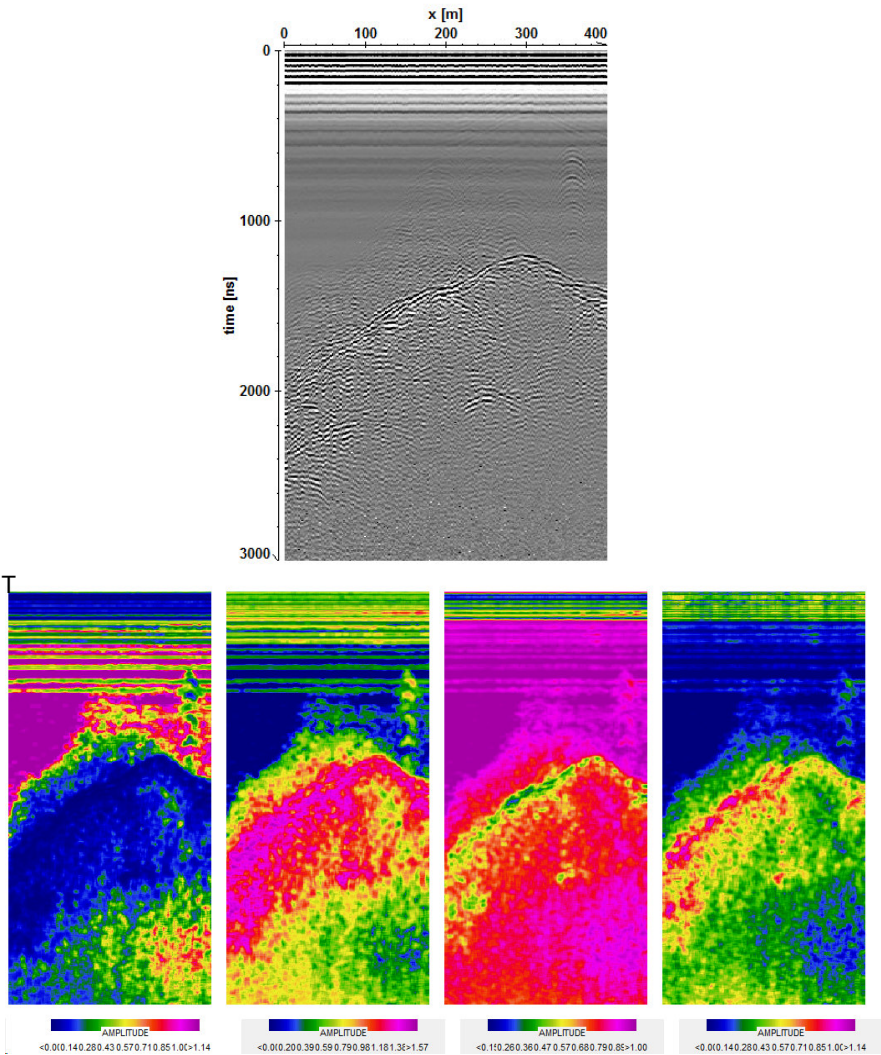
Four different texture attributes may be calculated:

uniformity: it reflects the overall uniformity (homogeneity) of the gray level distribution. Sometimes it is also named energy.

entropy: it is a measure of the overall uncertainty (randomness)

local homogeneity: it is a measure of the local homogeneity

local dissimilarity: it is used to detect the local dissimilarity (contrast)



1.11.6 Trace Interpolation/Resorting

The following trace interpolation and resorting possibilities are included within the trace interpolation/resorting group:

- markerinterpol
- traceincr-resampling
- make equidist.traces
- fix profile length
- split file
- YFlipProfile
- XFlipProfile
- resort-traceheader
- resort-group
- Yo-Yo section
- VSP corridor stack
- Traceinterpol-3DFile
- Xflip-3DFile
- shift-3DFile
- create 3D-ensembles

When activating the option `markerinterpol` a table appears. With this option the inputs may be entered either interactively within the original profile or using the table input.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number**.

The filter parameters for the individual processing steps are entered within the **filter parameter group**. The up/down arrows near the input fields allow to change the actual entered value by + or - 10 %. With the option **Apply on example trace** activated the effect of the filter parameters is online controlled when changing the parameters by showing both the original and filtered trace. The arrows near the input fields may be best used for such an interactive control. Deactivate this parameter if this control takes too much computer time.

After having specified the filter parameter you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing.

Activating the option **sequence proc.** additionally open the Sequence Processing menu. With this menu opened you may easily add the current processing step to the sequence processing flow which may be applied to a choosable number of lines after.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing. Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing.

1.11.6.1 Markerinterpol

The option allows an interpolation of the data in X-direction on the basis of markers to be set manually or to be automatically extracted from the data (suboption ReadMarker).

After having activated the option a table appears which allows the interactive input of the marker positions in traces (**trace nr.**) and the distance between successive markers (**marker inc.**). The program recalculates the number of traces between two set markers based on the wanted **trace increment** and the current set distance between the two markers (marker inc.) using the following formula: calculated trace numbers = marker inc. / trace increment. The **marker increment** must be predefined within the filter parameter group and can be individually changed for each marker position within the table input.

The trace increment must be a multiple of each marker increment. This is due to the fact that the number of traces within each marker increment must be rounded. The program automatically calculates the total marker range and determines the wanted number of traces based on the current trace increment. If this number does not correspond to the current trace number got from the filter markerinterpol a warning message appears.

The marker positions may be defined either within the original profile using the left mouse key or by entering a trace number within the table input. Individual marker increments must be entered within the table input. The 3. column of the table shows the cumulated markerdistance (cannot be changed). In addition it is possible to read the marker positions from the trace header of the original file (option **ReadMarker**). With the option **comment marker** activated the comment markers are loaded in addition (see also edit comment markers). If the markers at the first and last trace are missing you may use the option insert markers within the Edit several FileHeaders menu (chap. 1.10) . If the option is used within the Sequence Processing the marker positions are always read from the trace headers of the individual original file. With activated option comment marker the distance and the comment markers are used, with deactivated option only the distance markers are used. The parameter **SkipTraces** allows to avoid that double marker clicks are considered. After one marker position was found the next marker to be found must be at least SkipTraces away. The option **calc.markerincr.** allows to calculate the individual markerincrements based on the current traceincrement and number of traces between the markers. With the option **first trace fix** and/or **last trace fix** activated the range before the first and after the last marker will be assumed to have the same scale as the range between the first and second marker as well as the last and second to last marker. The markerincrement within these two ranges will be determined automatically from the entered markerincrements and the scale of the two ranges. Activating these options is useful if the profiles have arbitrary start and end points but special first and last marker positions. These options are also used for the sequence processing. With the option **keep all dist.markers** activated the distance markers which have not been used for the marker interpolation will be kept within the processed file. If deactivated the non used distance markers will be lost within the processed file. In the latter case the resulting distance markers do exactly represent the marker intervals.

With the option **update actual picks** activated the actually loaded picks will be relocated corresponding to the normal profile and shown as secondary picks both within the primary and the secondary processed file. After having performed the processing step the pick save menu opens which allows to store the new relocated picks. The option is only enabled if picks have been loaded and the option is not available within the sequence processing.

The option **save** stores the current edit values on an ASCII file under the project directory with a choosable name and the extension MAR. Each line of the ASCII file consists of two columns: 1. column: trace number, 2. column: marker increment. With activated option comment marker the comments of the comment markers are stored in addition within the 3. column.

The option **load** loads the wanted edit values from an ASCII file with the filename extension MAR. The table and the interactive window are automatically updated. With the suboption **ignore tracennr.** activated the tracenumbers within the ASCII-file will not be read in. This might be useful for example if parallel lines have been acquired and the markers have been set at identical not equidistant positions within the profiles. With the suboption **view only** activated the markers from the ASCII-file are only displayed with red symbols. It is possible to enter a shift value for the position of the first marker relative to the current profile. This option might be useful to control the current marker positions with an external relocation file in order to check if some markers are missing or false. Of course the option is only useful if the distance scale of the profile already fits quite well. The option **reset** resets the current edit values.

The option **batch save** generates ASCII markerfiles for a series of profiles for a subsequent sequence processing (option read from profile markerfile). The filenames of the ASCII-file correspond to the profile filename (without extension) and are stored directly under the projectdirectory with the extension mar. The actual markerincrement is used. This markerincrement can be changed externally using any editor program in order to use nonequidistant markerincrements within the sequence processing.

The program determines the corresponding number of traces between all set markers after the selection and executes a recalculation of the traces between successive markers. If more traces within the original profile between successive markers are available, some traces are omitted. If fewer traces are available, some traces are added within each marker part until the wanted number of traces is reached (the traces are simply added and not interpolated between successive original traces as such an interpolation may significantly change the shape of the onsets). The data before the first marker and behind the last marker are ignored, i.e. the program assumes that the beginning and the end of the profile are always indicated by a marker.

The trace with the last marker position is also included. Therefore, the total number of traces is increased by 1.

An example shall demonstrate the proceeding:

You have gained a profile of 0 to 10 meters with a particular scan rate (time). There, 1200 traces were recorded. The program has calculated an average trace interval of 0,00833 METERS from the distance specifications and the number of recorded traces. The marker interval, i.e. the distance of the marker indications recorded with the data in the field is 1 meter. You want to make the profile

spatially equidistant, now. The desired trace interval after making equal intervals, i.e. the distance between two subsequent traces, shall be 0,01 METERS. This value is in principle independent of the current average trace interval. If you choose a smaller value the data volume in total is enlarged. For a larger value than the current average trace interval the data volume is decreased accordingly. You choose the option `markerinterpol` and specify for the marker increment the value 1 METER and 0.01 METERS for the trace increment. Then you select on the basis of the markers recorded with the data a total of 11 markers with the mouse (indicate beginning and end of the profile - therefore 11 markers). After confirmation of the selection and specification of the processing label the profile is made equidistant. The result is a total of 1000+1 traces with a trace interval of 0,01 METERS.

The specification of the trace interval is determined by the lateral resolution capacity, i.e. the investigation depth and the frequency spectrum of the source. It is appropriate to find the best compromise between the smallest possible data volume and necessary spatial sampling rate. Of course you can also subsequently modify the spatial sample interval, e.g. with the options `expand`, `compress` or `stack traces`.

Peculiarity for GSSI data recorded from older analogue apparatuses:

The markers are placed on the data itself. Therefore, the option **ReadMarker** cannot read the marker positions directly from the trace header. In this case the option `ReadMarker` allows the automatic recognition of markers directly from the data, if they show certain properties. The program determines within a predefinable time window (**1. time** and **2. time** within the filter parameter group) the average amplitude change in x-direction from trace to trace and also the average total amplitude. In addition the amplitude change from to the next trace but one is determined. If one of these two amplitude changes exceeds the average change value multiplied with a scaling factor (option **scale**) within this time range and the number of extrema within the predefined windows reaches the value of **extrama** a marker is recognized. The automatically recognized markers are displayed and can be reedited manually.

When using the processing step `markerinterpol` within the **sequence processing**, the marker positions are always read from the trace headers of the individual original file. You may choose for the markerincrements between:

1. option `equidistant` - the entered single marker increment is used for all marks.
2. option `read from one markerfile` - the markerincrements are read from one single ASCII-file which has been created using the option `save` under `markerinterpol`. The markerincrements are the same for all profiles. The tracenumbers which are also stored within the `mar` file are of no importance because the marker positions are always read from the traceheader of the individual original file.

3. option `read from profile markerfile` - the markerincrements are read from a separate ASCII-file for each profile. The tracenumbers which are stored within the `mar` file are used for the marker positions (in contrast to item 2). The filename of the ASCII-file corresponds to the profile filename (without extension) and must have been stored directly under the `projectdirectory` and must have the extension `mar` (see also option `batch save`). Example for 2 profiles:

```
c:\data\procddata\file01.00t  C:\data\file01.mar  
c:\data\procddata\file02.00t  C:\data\file02.mar
```

1.11.6.2 Traceincr-resampling

The option performs a resampling of the data in x-direction for the given new trace increment.

The filter parameter **trace increment** specifies the new trace increment.

The radio box **traceincr-resampling** controls how the resampling will be done.

With the option **fix tracenummer** activated the profile will not be changed - only the actual traceincrement will be replaced by the new trace increment. This is the same as changing the traceincrement within the fileheader.

With the option **fix profile length** activated the profile will be changed. If the new number of traces resulting from the new trace increment is smaller than the trace number of the original file, some traces are omitted. If the new number of traces resulting from the new trace increment is larger than the trace number of the original file, some traces are added. In this case the traces are simply added and not interpolated between successive original traces as such an interpolation may significantly change the shape of the onsets. If you want the traces to be interpolated, you must use the option expand.

This option might be useful if data with different trace increments are to be combined.

1.11.6.3 Make equidist.traces

The option allows to interpolate non-equidistant data in such a way that the resulting data are equidistant. The precondition is that the true position of each trace is stored in the parameter **distance** within the individual trace header.

The non-equidistant data are resampled in x-direction based on the filter parameter **trace incr.** and the distance values stored in the individual trace headers of each trace. In addition the **start distance** and the **end distance** (starting and ending position of the new profile) have to be specified in the given distance dimension. By default the start distance and the end distance are determined from the individual traceheaders. By the manual input you may extract a distinct part from the profile. When using within the sequence processing the start and end distance values will always be taken from the individual traceheaders.

The option **no interpolation** controls how the new traces will be generated:

With the option **no interpolation deactivated** the program searches for each new distance point the next traces at the right and the left and performs a linear interpolation between these two traces unless one of the two trace header positions exactly match the new one.

With the option **no interpolation activated** a new parameter **bin for stacking** must be entered. The program performs a horizontal stacking over all traces which lay within the bin range around each new distance point. If no traces are present within this range around one distance point the new trace only contains zero values. It is possible to interpolate these 0-value traces within a subsequent processing step named interpolate (chap. 1.11.7.3, suboption get 0 traces). This is also possible within the sequence processing.

With the option **use tracedelays** activated the individual traces are shifted based on the given traceheader delaytimes for the internal interpolation or stacking process. This may be useful if the data had not been corrected before for the individual trace timedelays.

As a result the data are equidistant in profile direction and all processing steps like stacking or fk-filtering may be applied on.

1.11.6.4 fix profile length

The option allows to fix the profile length to a given value.

The filter parameter profile length specified the new length of the profile in the given distance dimension. If the original length is greater than this value the corresponding traces will be omitted. In case of a smaller original profile length traces with zero amplitude values will be added.

1.11.6.5 split file

The option allows to split the original datafile into different independent files.

The splitting positions may be defined either within the original profile using the left mouse key or by entering a trace number within the table input. The options **read marker** and **read ensembles** read the splitting positions from the trace header of the original file (marker or ensemble number respectively). With the options **first trace marker** or **last trace marker** activated a splitting position will be automatically included at the first or last trace position of the profile. Activating this option is useful if no marker has been set at the start or end of the profile. A manual change is always possible. The first splitted file starts at the first splitting tracenummer (all traces before this tracenummer will not be considered) and ends one trace before the the next splitting tracenummer. The last splitted file starts at the last but one position and ends one trace before the last splitting position (all traces after (including) the last splitting tracenummer will be omitted).

After having started the option the overall filename must be entered. An automatic labelling is added to the filename. Example with 100 different independent files: entered new overall filename: test - the new filenames will be: test0001.00t, test0002.00t, ..., test0100.00t. The files are stored under the subdirectory prodata with the extension 00t.

The option might be useful for:

- splitting a long 2D-profile (datatype: const.offset) into different partions (datatype: const.offset). In this case the profile start coordinate within the fileheader will be taken from the startposition within the original file.
- splitting a 3D-file (datatype: 3D-const.offset) into different parallel 2D-lines (datatype: const.offset): In this case the profile startcoordinate will be the same for all 2D-lines, the profile constant coordinate will be changed based on the given lineincrement.
- splitting a several shot file (datatype: several shots) into different shots (datatype: single shot): In this case the shot position as well as the start and end receiver position of the splitted shots will be taken from the traceheaders of the original file.

The option is available within the sequence processing.

1.11.6.6 YFlipProfile

The option allows to flip the data in time-(y-)direction.

The first sample will be the last one and the last sample will become the first sample. The starting time will be unchanged. The option might be used if the data samples are not correctly sorted during the data acquisition.

1.11.6.7 XFlipProfile

The option gives the possibility to flip a profile in x-direction.

For the inversion the program uses the RAM-memory if the number of traces is smaller/equal 8192. If the profile will not be loaded into the kernel memory of the computer because of insufficient RAM-memory, the option is carried out using the hard disk (this may take some time).

The beginning and end coordinates in the profile direction are exchanged after execution of the option if the beginning coordinates are larger than the end coordinates.

To be considered: If the traceheader coordinates shall be updated based on an ASCII-file (e.g. GPS coordinates) the option XFlipProfile should always be applied after having updated the traceheaders because the original tracenumbers mostly form the base for the traceheader updating.

1.11.6.8 Resort-traceheader

The option allows to resort the data in x-direction based on the distances or on the collected times or on the ensemble no stored in the individual trace headers.

With **distancies** activated within the “sorting by” box the option allows the resorting in x-direction of the data based on the current traceheader distances. The precondition is that the true position of each trace is stored in the parameter **distance** within the individual trace header. After sorting the first trace corresponds to the minimum distance and the last trace to the maximum one.

With **times collected** activated within the “sorting by” box the option allows the resorting in x-direction of the data based on the current traceheader TimeCollect values. The precondition is that the true time collect of each trace is stored in the parameter **Time Collect** within the individual trace header. After sorting the first trace corresponds to the minimum time and the last trace to the maximum one.

With **ensemble no** activated within the “sorting by” box the option allows the resorting in x-direction of the data based on the current traceheader ensemble number. After sorting the first trace corresponds to the minimum ensemble number and the last trace to the maximum one.

1.11.6.9 Resort-group

The option allows a resorting of the data for a given trace group.

The two filter parameter nr. of traces and trace difference must be entered. **Nr. of traces** gives the number of traces which shall be combined. **Trace difference** gives the trace number difference of the traces to be combined. If any of the two parameters is set to 0 the parameter will be automatically calculated from the total number of traces divided by the other parameter. This might be useful if the processing step shall be used within the sequence processing with different number of traces.

Example1: There are 48 traces. Resorting shall be done in the following way: traces 1,13,25 and 37 are written on position 1,2,3 and 4; traces 2,14,26 and 38 on position 5,6,7 and 8 and so on. The following inputs are necessary: trace number: 4; trace incr.: 12.

Example2: There are two shot lines within one file. Each shot line consists of 120 shots with 24 channels and a receiver increment of 2 m. Two shots of each line shall be combined together because the receiver positions are the same and the shot receiver positions differ by 1m (1. shot line: first receiver at 1 m from shot position; 2. shot line: first receiver at 2 m from shot position). The 1. shot gather range from trace nr 1 to 2880 and the second one from trace n. 2881 to 5760. Resorting is done by entering: nr of traces = 2, trace difference = 2880. After resorting one complete shot gather results with a total amount of 120 shots and 48 channels with a receiver increment of 1m and the first receiver at 1 m).

One useful application is the possibility of putting together different shot lines (see example 2). For this you must create the two different lines using the conversion sequence combine lines/shots under Import ConversionMode and then combine these two lines within one single file using the option insert profile (attach). This combined file may be resorting in the way described above.

1.11.6.10 XY-exchange

The option allows an exchange of the sorting in x-(distance)- and y-(time-) direction. The option might be useful if the original data have not been sorted as Reflexw is assuming, e.g. x-values along the y-axis or the time-values (samples) along the x-axis.

1.11.6.11 Yo-Yo section

The option allows the separate presentation of downhole-crosshole reflection data (source and receiver hole) coming from below and from above a predefined depthpoint.

Only those shot-receiver combinations are sorted out which direct wave is going through a choosable depth midpoint resulting in a so called Yo-Yo section. In constant velocity media Yo-Yo reflections from an interface have constant path lengths. Therefore any horizontally layered reflector will appear on a Yo-Yo section as a flat one.

The dataset must consist of different shots each containing the same number of equidistant traces (receivers). The total number of traces within the dataset is the number of shots multiplied with the number of receivers. The receivers as well as the shots must be sorted with increasing depths. The shot and receiver coordinates must be defined within the traceheader.

First the data are automatically separated into two datasets one containing the upward travelling waves and the other one containing the downward travelling waves. For that purpose a fk-filtering is used. The filtering is done for each shot independently. For that purpose the complete dataset is subdivided into n different subsets each containing equal number of receiver traces (option trace number). Afterwards the data are sorted around the chosen midpoint.

The sorting depends on the entered x-coordinate of the shot and the receiver. If the shot x-coordinate is smaller than the receiver x-coordinate the shot x-coordinate is used as the starting coordinate of the resulting yo-yo section and vice versa.

The option needs the following inputs:

midpoint: defines the midpoint depthpoint for creating the Yo-Yo section. Only these shot-receiver combinations are taken into account which direct wave is going through this midpoint. (within the range of the chosen binsize).

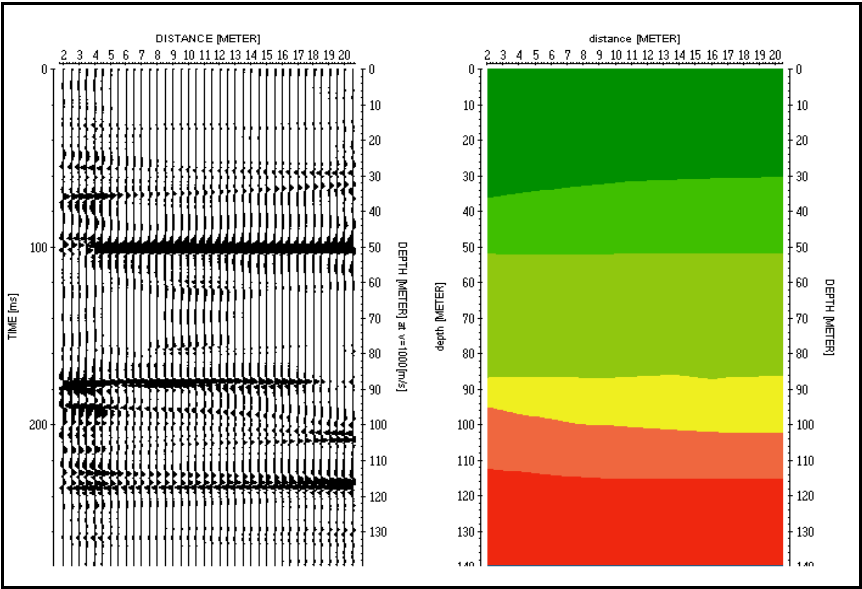
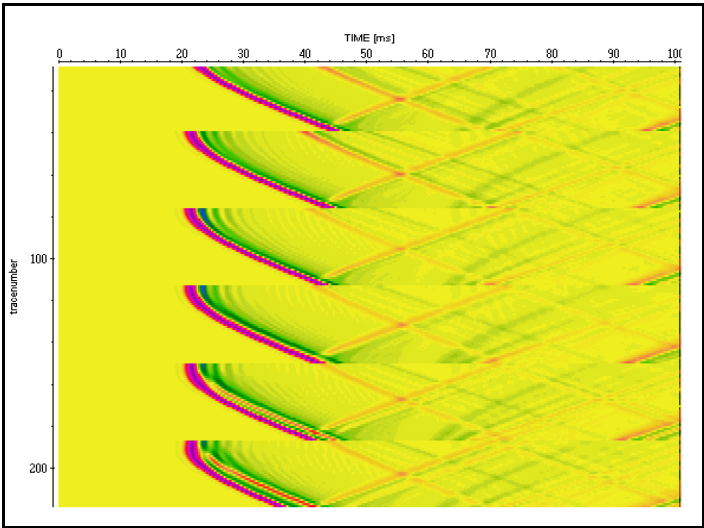
binsize: defines the range for cutting the midpoint (see above). A value not equal 0 is necessary if source and receiver coordinates and the midpoint coordinate do not exactly match.

trace number: enter the number of traces per shot. The input is necessary for the fk-filtering of the individual shots.

velocity: enter the velocity which is used for calculating the traveltime of the direct wave from the shot to the receiver. This traveltime is subtracted from the traveltime of each trace.

The following pictures show an example with 41 shots between 50 and 90 m and 37 receivers between 52 and 88 m (picture on the left shows part of the original crosshole data - 6 shots).

The Yo-Yo section has been created around the midpoint 70 m with a binsize of 0.5 m, 37 number of traces and a velocity of 1000 m/s. On the right panel the model is shown.



1.11.6.12 VSP corridor stack

The option allows the dynamic correction and stacking of the reflections of a vertical seismic profile (VSP). The result is a time-shifted two-way arrival times (at the surface) section.

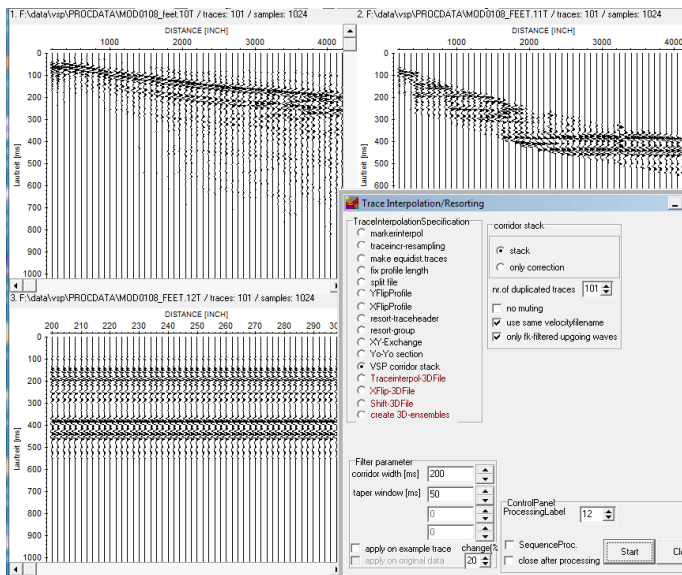
The summation is done based on the first arrival timeshift over the **corridor width** given in the timedimension. The first arrivals used are calculated from the receiver positions and the 1D-velocity distribution which has been created before within the CMP-vel.analysis module. With the option **use same velocity filename** activated the 1D-velocity file which has the same filename like the original datafile will be automatically used if available. If not a velocity file will be queried.

The data are muted above the first arrivals and below the time beginning at the first arrival time plus the corridor width. A taper may be applied around the first arrivals (option **taper window**).

With the option **only fk-filtered outgoing waves** activated a fk-filtering will be done before the stacking/correction resulting in a section which only contains the outgoing wavefield.

It is possible to perform a stacking (picture at the bottom) or only a time shift correction (picture at the right) of the data (option **stack** or **only correction** activated). If a stacking shall be performed the **number of duplicated traces** for the stacked section may be entered.

The number of samples does not change. If the original data acquisition time is small and deeper reflections are present you may use the option time cut before in order to extend the timerange of the original profile.



1.11.6.13 Traceinterpol-3DFile

The option allows to reinterpolate each 2D-line of a 3D-file in x-direction.

A 3D-file consists of several 2D lines each separated by a constant increment (see also line increment under fileheader coordinates). The 2D lines are stored successively within the 3D file. Normally the 2D lines should have the same length and the same number of traces. If the number of traces within each 2D-line differs, this option serves to compensate for these irregularities.

You must enter the two filter parameters trace increment and profile length in the given distance dimension. **Trace increment** specifies the wanted trace increment after interpolation. **Profile length** specifies the length of each 2D profile line of the 3D file. The program recalculates the number of traces within each 2D-line based on the wanted **trace increment** and **profile length** using the following formula: $\text{calculated trace numbers} = \text{profile length} / \text{trace increment}$.

Use this option if the single 2D-lines within a 3D-file don't have the same number of traces because of some irregularities during the data acquisition (e.g. slip of the measure wheel).

1.11.6.14 XFlip-3DFile

The option gives the possibility to flip the individual ensembles of a 3D-file in distance-direction.

You must enter the **Flip mode** within the filterparameter group. Enter 0 if all ensembles within the 3D-files shall be flipped. Enter 1 if every 2. ensemble shall be flipped starting at ensemble number 2. Enter 2 if every 2. ensemble shall be flipped starting at ensemble number 1. The values 1 or 2 are useful if the data have been acquired meandering.

1.11.6.15 Shift-3DFile

The option gives the possibility to shift each or every 2. ensemble for a definable range.

The **shift mode** defines which ensembles shall be shifted. Enter 0 if every ensemble shall be shifted. Enter 1 for shifting every 2. ensemble starting at ensemble number 2 and enter 2 for shifting every 2. ensemble starting at ensemble number 1. The **shift range** in the given distance dimension defines the range for the shift of each 2. (or each) ensemble.

For example the option may be useful if a systematic shift of the traces has been occurred during the meandering acquisition of a 3D-datafile.

1.11.6.16 create 3D-ensembles

This option allows you to subdivide the current profile into different 2D-lineparts. This option allows you for example to redefine the individual ensembles (2D-lines) of a 3D-file (the Ensemble-Nr. within the traceheaders will be updated).

With the suboption **automatic** activated you only have to enter the number of traces per linepart (filter parameter: **nr.of traces**). In this case the current profile is subdivided into equidistant 2D-lineparts each containing the given number of traces.

With the suboption **manual** activated you must manually enter the start position of each linepart within the table or using the left mouse button within the profile.

1.11.7 Edit traces/traceranges

The following edit possibilities are included within the Edit traces/traceranges group:

- remove
- interpolate
- extract
- replace
- reverse polarity
- set to zero
- duplicate
- move
- swap 2 timeblocks
- insert profile
- combine files f.CMP
- merge files
- merge in timedir.
- insert zero traces
- remove zero traces
- remove idle traces

When activating the options remove, interpolate, extract, replace, reverse polarity, set to zero, duplicate and move a table appears. With all these options the inputs may be entered either interactively within the original profile or using the table input.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number**.

The filter parameters for the individual processing steps are entered within the **filter parameter group**. The up/down arrows near the input fields allow to change the actual entered value by + or - 10 %. With the option **Apply on example trace** activated the effect of the filter parameters is online controlled when changing the parameters by showing both the original and filtered trace. The arrows near the input fields may be best used for such an interactive control. Deactivate this parameter if this control takes too much computer time.

After having specified the filter parameter, you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing.

Activating the option **sequence proc.** additionally opens the Sequence Processing menu. With this menu opened you may easily add the current processing step to the sequence processing flow which may be applied to a choosable number of lines after.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing.

Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing. Exceptions where only one secondary line will be needed :

processing step **insert profile** and **replace**: here an additional character(s) is asked which will be automatically added to the actual filenames for the creation of the secondary lines which shall be inserted or replaced. Example:

actual filename: data_gps.00t - additional characters: _insert -> secondary file: data_gps_insert

1.11.7.1 Remove

This option allows to delete individual traces or trace ranges.

After having activated the option **remove** a table with the parameters **1.trace nr.** and **2. trace nr.** which allows the interactive input of the individual trace ranges. If the trace range consists of only one trace 1.trace nr. and 2.trace nr. must have the same value.

The traces (trace ranges) can also be defined within the original profile. With the option **EditTraces** activated you may select the desired trace by pressing the left mouse button at the desired position. With the option **EditTraceRanges** activated you may select the desired trace range by clicking on the uppermost left corner of the area and dragging the mouse to the wanted lowermost right corner with the left mouse button pressed. The chosen trace numbers are automatically transferred to the table input and vice versa to the profile image.

The option **save** stores the current edit values on an ASCII file under the project directory with the name **remove** and a freely choosable extension. Each line of the ASCII file consists of two columns: 1. column: 1. trace nr., 2. column: 2. trace nr.. If used within the sequence processing such a file will be asked for.

The option **load** loads the wanted edit values from an ASCII file with the filename **remove**. The table and profile image are automatically updated.

The option **reset** resets the current edit values.

With the option **use line parts** activated the entered traces are used for all lineparts (e.g. of a 3 D-file or a several shots file).

With the option **keep markers** activated any marker will be kept if it falls into the range to be removed.

The number of traces in the processed file decreases according to the defined trace ranges. Please note that the trace headers are maintained for all remaining traces. This mainly concerns the coordinates. A shifting of the distance coordinate according to the number of deleted traces does not occur.

1.11.7.2 Remove range

This option allows to delete a range of traces at the beginning and at the ending of the profile.

The filter parameter **front range** and **back range** determine the size of the trace ranges (in the given distance dimension) to be removed. Enter a size of 0 for the corresponding parameter if you do not wish to remove a range either at the beginning or at the ending. The filter parameter **upd.coord.** allows you to determine if the starting and ending coordinates in profile direction are automatically updated. Enter a value different from 0 if you do not wish the start coordinates to change.

1.11.7.3 Interpolate

The option allows to replace individual traces or trace ranges with an interpolation of the two neighbouring traces. The interpolation is performed over samples with same time. When selecting marginal traces the corresponding neighbouring trace is duplicated.

After having activated the option **interpolate** a table with the parameters **1.trace nr.** and **2. trace nr.** which allows the interactive input of the individual trace ranges. If the trace range consists of only one trace 1.trace nr. and 2.trace nr. must have the same value.

The traces (trace ranges) can also be defined within the original profile. With the option **EditTraces** activated you may select the desired trace by pressing the left mouse button at the desired position. With the option **EditTraceRanges** activated you may select the desired trace range by clicking on the uppermost left corner of the area and dragging the mouse to the wanted lowermost right corner with the left mouse button pressed. The chosen trace numbers are automatically transferred to the table input and vice versa to the profile image.

The option **save** stores the current edit values on an ASCII file under the project directory with the name **interpol** and a choosable extension. Each line of the ASCII file consists of two columns: 1. column: tracenummer1, 2. column: range. The option **load** loads the wanted edit values from an ASCII file with the filename **interpol** and any extension. The table and the interactive window are automatically updated.

The option **reset** resets the current edit values.

The option **get 0 traces** allows the interpolation of those traces which only contain 0 values.

The option **generate per.groups** allows the interpolation of periodic groups of traces. One possible application is the elimination of groups of traces disturbed by near surface periodic metal objects to be repeated periodically (e.g. rails). Before activating this option the following filter parameter within the filter group box must be entered: **start distance** and **end distance** define the distance range for the interpolation. In addition start distance characterizes the begin of the first group of traces to be interpolated. **Group interval** defines the distance between the single groups and **group length** defines the length of each individual group. All inputs are in the given distance dimension. After activating the option **generate per.groups** the current table inputs are reset and recalculated based on the current filter parameters.

Only the interpolation of 0 traces or periodic groups based on the set filter parameters is supported within the **Sequence Processing**. If the entered end distance is smaller or equal the start distance the interpolation is done for 0 traces. Otherwise the interpolation is done on periodic groups.

1.11.7.4 Extract

The option allows to extract individual traces or trace ranges, respectively, from the primary file.

After having activated the option **extract** a table with the parameters **1.trace nr.** and **2. trace nr.** which allows the interactive input of the individual trace ranges. If the trace range consists of only one trace 1.trace nr. and 2.trace nr. must have the same value.

The traces (trace ranges) can also be defined within the original profile. With the option **EditTraces** activated you may select the desired trace by pressing the left mouse button at the desired position. With the option **EditTraceRanges** activated you may select the desired trace range by clicking on the uppermost left corner of the area and dragging the mouse to the wanted lowermost right corner with the left mouse button pressed. The chosen trace numbers are automatically transferred to the table input and vice versa to the profile image.

The option **save** stores the current edit values on an ASCII file under the project directory with the name **extract** and a freely choosable extension. Each line of the ASCII file consists of two columns: 1. column: 1. trace nr., 2. column: 2. trace nr.. The option **load** loads the wanted edit values from an ASCII file with the filename **remove**. The table and profile image are automatically updated.

The option **reset** resets the current edit values.

The option **read marker** loads all traces including a marker. The table and profile image are automatically updated.

The option **from xy-tracecoord.** allows the automatic determination of the wanted traces from the traceheader xy-coordinates. Before activating this option the xy-range must be entered within the filter group box: **start x-coord.** and **start y-coord.** as well as **end x-coord.** and **end y-coord.**

With the option **extract coordinate/time range** activated the coordinate/time range defined with filterbox will be used for the data extraction. The profile start coordinate will be shifted according to the coordinate of the entered start coordinate of the extraction range. The start time will be set to the entered start time of the extraction range.

With the option **use line parts** activated the entered traces are used for all lineparts (e.g. of a 3 D-file or a several shots file).

With the option **shift profile start coordinate** activated the profile start coordinate will be shifted according to the coordinate of the first chosen trace number. Otherwise the fileheader coordinates remain unchanged except for a 3D-file (option **datatype** set to 3D-const.offset within the fileheader). In this case the start coordinates in x- and y-direction will be changed if the start x-coord. and/or the start y-coord. within the filter group box are larger than the corresponding values stored within the fileheader.

The trace headers remain. The extracted traces or trace ranges are saved trace-oriented sequentially in increasing order.

If used within the Sequence Processing the traces are always automatically determined from the traceheader coordinates (see option from xy-tracecoord.).

1.11.7.5 Replace

The option allows to replace individual traces or trace ranges at equal traces by the corresponding traces of another file.

After having activated the option **replace** a table with the parameters **1.trace nr.** and **2. trace nr.** which allows the interactive input of the individual trace ranges. If the trace range consists of only one trace 1.trace nr. and 2.trace nr. must have the same value.

The traces (trace ranges) can also be defined within the original profile. With the option **EditTraces** activated you may select the desired trace by pressing the left mouse button at the desired position. With the option **EditTraceRanges** activated you may select the desired trace range by clicking on the uppermost left corner of the area and dragging the mouse to the wanted lowermost right corner with the left mouse button pressed. The chosen trace numbers are automatically transferred to the table input and vice versa to the profile image.

The option **load** within the SecondLineGroupBox enables the opening of the file from which the traces to be replaced should be taken. The replacement is performed at equal traces, that is the data at the specified trace positions or trace ranges, respectively, are replaced by the data of the new file at the identical trace positions.

The option is not available within the Sequence Processing.

The option is available for the **batch start** processing. Here an additional character(s) is asked which will be automatically added to the actual filenames for the creation of the secondary lines which shall be inserted. Example: actual filename: data_gps.00t - additional characters: _insert -> secondary file: data_gps_insert

1.11.7.6 Reverse polarity

The option allows to reverse the polarity of individual traces or trace ranges, this means a phase shift by 180 degrees.

After having activated the option **reverse polarity** a table with the parameters **1.trace nr.** and **2. trace nr.** which allows the interactive input of the individual trace ranges. If the trace range consists of only one trace 1.trace nr. and 2.trace nr. must have the same value.

The traces (trace ranges) can also be defined within the original profile. With the option **EditTraces** activated you may select the desired trace by pressing the left mouse button at the desired position. With the option **EditTraceRanges** activated you may select the desired trace range by clicking on the uppermost left corner of the area and dragging the mouse to the wanted lowermost right corner with the left mouse button pressed. The chosen trace numbers are automatically transferred to the table input and vice versa to the profile image. With the option **use line parts** activated the entered traces are used for all lineparts (e.g. of a 3 D-file or a several shots file). This might be useful especially for a multi shot file where e.g. one geophone of a moving line always exhibits the wrong polarity.

The option **save** stores the current edit values on an ASCII file under the project directory with the name reverse and a choosable extension. Each line of the ASCII file consists of two columns: 1. column: tracenumber1, 2. column: tracenumber2. If used within the sequence processing such a file will be asked for.

The option **load** loads the wanted edit values from an ASCII file with the filename reverse and any extension. The table and the interactive window are automatically updated.

The option **reset** resets the current edit values.

The option is also available within the Sequence Processing.

1.11.7.7 Set to zero

The option allows to set individual traces or trace ranges to zero.

After having activated the option **set to zero** a table with the parameters **1.trace nr.** and **2. trace nr.** which allows the interactive input of the individual trace ranges. If the trace range consists of only one trace 1.trace nr. and 2.trace nr. must have the same value.

The traces (trace ranges) can also be defined within the original profile. With the option **EditTraces** activated you may select the desired trace by pressing the left mouse button at the desired position. With the option **EditTraceRanges** activated you may select the desired trace range by clicking on the uppermost left corner of the area and dragging the mouse to the wanted lowermost right corner with the left mouse button pressed. The chosen trace numbers are automatically transferred to the table input and vice versa to the profile image.

The option **save** stores the current edit values on an ASCII file under the project directory with the name setzero and a freely choosable extension. Each line of the ASCII file consists of two columns: 1. column: 1. trace nr., 2. column: 2. trace nr.. If used within the sequence processing such a file will be asked for.

The option **load** loads the wanted edit values from an ASCII file with the filename correct. The table and profile image are automatically updated.

The option **reset** resets the current edit values.

1.11.7.8 Duplicate

The option allows to duplicate individual traces.

After having activated the option **duplicate** a table with the parameters **1.trace nr.** and **2. trace nr.** which allows the interactive input of the individual trace ranges. If the trace range consists of only one trace 1.trace nr. and 2.trace nr. must have the same value.

The traces can also be defined within the original profile. With the option **EditTraces** activated you may select the desired trace by pressing the left mouse button at the desired position. The chosen trace numbers are automatically transferred to the table input and vice versa to the profile image.

The number of traces in the processed file increases according to the number of selected traces.

The option is not available within the sequence processing. The option is available for the batch start processing.

1.11.7.9 Move

The option allows to move individual traces in time(y)-direction in order to balance possible trigger errors. This movement corresponds to a static correction.

After having activated the option **move** a table with the parameters **trace nr.** and **time** in the given timedimension which allows the interactive input of the individual trace ranges. If the trace range consists of only one trace 1.trace nr. and 2.trace nr. must have the same value.

The traces (trace ranges) can also be defined within the original profile. Choose the desired trace by the left mouse and move it in time direction with pressed mouse key. The chosen trace number and the moving time are automatically transferred to the table input and vice versa to the profile image.

The option **save** stores the current edit values on an ASCII file under the project directory with the name move and a freely choosable extension. Each line of the ASCII file consists of two columns: 1. column: trace nr., 2. column: time shift.. If used within the sequence processing such a file will be asked for.

The option **load** loads the wanted edit values from an ASCII file with the filename correct. The table and profile image are automatically updated.

The option **reset** resets the current edit values.

1.11.7.10 Swap 2 timeblocks

The option allows to swap two datablocks in timedirection. Example: the trace consists of 4096 samples. After having applied the option the timeblock of sample 1 to 2048 and the timeblock of sample 2049 to 4096 will be swapped, this means old sample 1 will be located at sample 2049 and old sample 2049 at position 1 and so on. Use this option if some traces are swapped for some reasons.

1.11.7.11 Insert profile

The option allows to insert a second file at a distinct position.

The options **EditTrace**, **Attach at end** and **use coordinates** control the position of the second file to be placed.

With the option **EditTrace** activated you may specify any trace number within the table input or by clicking within the original profile. This trace number defines the position of the second line to be placed.

Activating the option **Attach at end** means that combining is done at the end of the first profile independently from the original starting position of the second file.

Activating the option **use coordinates** means that combining is done at the starting position of the second file. One application is the combination of two files, which do have different ending and starting coordinates (e.g. because of a

non measurable area). The non measured area will be filled up with zero traces. If the profiles overlap the overlapping traces of the original profile will be put at the end of the processed file. If this is not desired use the option merge files (chap. 1.11.7.13) with the suboption attach profiles instead.

The option **load** within the SecondLineGroupBox enables the opening of the file which is to be inserted.

The resulting number of traces will be increased depending on the starting coordinate of the second profile if the option use coordinates is activated or on the number of traces of the inserted profile with the option EditTrace or Attach at end activated.

The option is not available within the **Sequence Processing**.

The option is available for the **batch start** processing. Here an additional character(s) is asked which will be automatically added to the actual filenames for the creation of the secondary lines which shall be inserted. Example: actual filename: data_gps.00t - additional characters: _insert -> secondary file: data_gps_insert

1.11.7.12 Combine files f.CMP

This option gives the possibility to combine together the original line with several secondary lines into one new file.

The options single shots and zero offset lines control the data type of the lines to be CMP combined and therefore the sorting of the traces within the new line. With the option **single shots** activated all data files are read in subsequently and written out. The first file is the original line, the second one the first secondary file and so on. For example this option serves for a combination of several shot gathers analogous to the option combine lines/shots within the Import format specification.

With the option **zero offset lines** activated the combining of the lines is done at the position of each trace. This option is necessary for a subsequent CMP-processing of constant-offset lines. Example: you want to combine 4 profiles trace by trace: After finishing the option the traces are sorted as followed: 1. trace of the original profile, 1. trace of the 1. secondary file, 1. trace of the 2. secondary file, 1. trace of the 3. secondary file, 2. trace of the original file, 2. trace of the 1. secondary file and so on. The trace increment and the time increment as well as the number of traces of each profile should be identical. The geometry (shot-, receiver and CMP-coordinates) are automatically updated based on the coordinates (profile coordinates and S/R-distance - see also fileheader coordinates and fileheader input1) stored in the profile header of each profile.

The option **load** within the SecondLineGroupBox enables the opening of the secondary files to be combined. The files are automatically sorted. Therefore, the order of choosing the files is of no importance.

The option `combine files f.CMP` is useful if the data shall be processed within the CMP-processing afterwards.

The option is not available within the Sequence Processing.

1.11.7.13 merge files

This option gives the possibility to merge the original line with several secondary lines into one new file.

The merging is done based on the start coordinates of the single lines stored within the individual fileheaders (see also fileheader coordinates). The traceincrement, the timeincrement and the starttime of each line must be identical.

Five different merging possibilities are given:

With the suboption **attach profiles** activated it is assumed that the profiles do not overlap and there might be gaps inbetween. The traceincrement of the resulting line will be the original traceincrement except for the case of all lines containing only 1 trace and an identical increment between the startcoordinates of the original files is given (e.g. a Finit Difference simulation of moving point sources and only one receiver). In this case the traceincrement is set to this increment. The resulting profile starts at the smallest startcoordinate and ends at the largest endcoordinate of the original profiles. The non measured area will be filled up with zero traces. The xy-traceheader coordinates within this gap will be interpolated from the given end xy-coordinates of the first profile and the start xy-coordinates of the second profile. If the profiles overlap only the overlapping traces of the newly attached profile will be kept (in contrast to the option `insert profile`, chap. 1.11.7.10).

With the option **merge inline** activated it is assumed that the startcoordinates of the different lines only differ by a fraction of the traceincrement. The traceincrement of the resulting line will be the original traceincrement divided by the number of the merged profiles.

example: 4 lines have to be merged together. Each line has a trace increment of 1 meter. The start coordinates of the individual lines differ by 0.25 meter.

original line: start coordinate: 0

1. sec. line: start coordinate: 0.25

2. sec. line: start coordinate: 0.5

3. sec. line: start coordinate: 0.75

The traces of the merged file are sorted as followed: 1. trace of the original profile, 1. trace of the 1. secondary file, 1. trace of the 2. secondary file, 1. trace of the 3. secondary file, 2. trace of the original file, 2. trace of the 1. secondary file and so on.

With the option **3D-files** or **3D-files meand** activated allows to merge the actually loaded 3D-file with different 3D-files which have been acquired e.g. using a multi antenna array or which have been acquired in order to condense the grid. Precondition is that each 3D-datafile consists of parallel 2D-lines with the same ensemble numbers for each set of 2D-lines and that the first 2D-line within the actually loaded 3D-datafile represents the start 2D-line.

Example four antenna array: the data have been acquired along parallel 2D-lines which have been stored within one single datafile for each antenna. The result are four different 3D-datafiles. Precondition for the use of the merge option is the existence of the correct (same) ensemble number for each 2D-line within each 3D-datafile. The resulting 3D-datafile after the merge option consists of the following ensembles:

1. Ensemble (2D-line) of the first 3D-datafile
1. Ensemble (2D-line) of the second 3D-datafile
1. Ensemble (2D-line) of the third 3D-datafile
1. Ensemble (2D-line) of the fourth 3D-datafile
2. Ensemble (2D-line) of the first 3D-datafile
2. Ensemble (2D-line) of the second 3D-datafile
2. Ensemble (2D-line) of the third 3D-datafile
2. Ensemble (2D-line) of the fourth 3D-datafile
- and so on

The option **3D-files meand.** allows a meandering sorting of the antenna array. The resulting 3D-datafile after the merge option consists of the following ensembles:

1. Ensemble (2D-line) of the first 3D-datafile
1. Ensemble (2D-line) of the second 3D-datafile
1. Ensemble (2D-line) of the third 3D-datafile
1. Ensemble (2D-line) of the fourth 3D-datafile
2. Ensemble (2D-line) of the fourth 3D-datafile
2. Ensemble (2D-line) of the third 3D-datafile
2. Ensemble (2D-line) of the second 3D-datafile
2. Ensemble (2D-line) of the first 3D-datafile
3. Ensemble (2D-line) of the first 3D-datafile
3. Ensemble (2D-line) of the second 3D-datafile
3. Ensemble (2D-line) of the third 3D-datafile
3. Ensemble (2D-line) of the fourth 3D-datafile
4. Ensemble (2D-line) of the fourth 3D-datafile
4. Ensemble (2D-line) of the third 3D-datafile
4. Ensemble (2D-line) of the second 3D-datafile
4. Ensemble (2D-line) of the first 3D-datafile

The profile increment will be automatically updated (divided by the number of 3D-datafiles).

With the suboption **stack profiles** activated the chosen profiles will be vertically stacked.

The option **load** within the SecondLineGroupBox enables the opening of the secondary files to be combined. Multiple choice is available using the shift or ctrl key.

The option is not available within the Sequence Processing.

1.11.7.14 merge in timedir.

This option allows you to merge different lines into one new file whereby the merging is done in time direction trace by trace. The data of one trace of all profiles are merged into one new trace. The new number of samples is the sum of the samples of the individual lines.

The option serves for multiplexing multichannel data which are automatically demultiplexed during the import. The number of traces for all lines to be merged should be equal as well as the time increment.

The profiles to be merged with the primary profile in time direction will be defined either automatically or by a manual choice. With **manual choice** activated the chosen profiles will be automatically sorted in alphabetic order with the primary file at the first position.

With **automatic** activated the program automatically searches for the files to be merged. This means that all datafiles with identical filenames except the last character which represents the channel number (see also Import filename specification, chap. 1.5.2) are assumed to be the multichannel data to be merged into one file.

Within the sequence processing only the automatic file choice is available.

1.11.7.15 insert zero traces

The option allows to insert traces with zero amplitude values.

After having activated the option **insert zero traces** a table appears with the parameters **trace nr.** and **range** which allow the interactive input of the position (trace nr.) and the range of zero traces to be inserted. The range must be entered manually for each position in the given distance dimension.

The traces can also be defined within the original profile. You may select the desired trace by pressing the left mouse button at the desired position. The chosen trace numbers are automatically transferred to the table input and vice versa to the profile image.

If the value for trace nr. is set to 0 a corresponding zero traces range is inserted before the first trace.

The option **save** stores the current edit values on an ASCII file under the project directory with the name insert0 and a choosable extension. Each line of the ASCII file consists of two columns: 1. column: tracenummer1, 2. column: range. If used within the sequence processing such a file will be asked for.

The option **load** loads the wanted edit values from an ASCII file with the filename insert0 and any extension. The table and the interactive window are automatically updated.

The option **reset** resets the current edit values.

1.11.7.16 remove zero traces

The option allows to automatically remove all traces with zero amplitude values. The traceincrement as well as the startcoordinate will be kept. The endcoordinate is updated based on the new number of traces.

1.11.7.17 remove idle traces

The option allows to automatically remove all adjacent traces which have equal or similar amplitude and phase, e.g. stemming from an idle state. You may enter the max. trace difference in percentage from the mean trace difference of all traces. The endcoordinate is updated based on the new number of traces.

1.11.8 2D-Filter

The following filter possibilities are included in the declipping/arithmetic functions group:

- running average
- subtracting average
- background removal
- compress
- expand
- stack traces
- subtract traces
- compress 3D-file
- expand 3D-file
- average xy-filter
- median xy-filter
- dilation xy-filter
- erosion xy-filter
- constrain borders
- average 3D-filter

For the filter background removal within this group the original trace is shown together with the currently filtered trace, the spectrum of the original trace and the spectrum of the currently filtered trace. Additionally the reference trace calculated from the filter input parameters is shown. This reference trace may also be saved for a subsequent viewing or preprocessing.

The arrows near the original trace and original spectrum are used to enhance the resolution of the trace and spectrum windows. The trace or the spectrum is stretched up and down when using these arrows. The spectrum scale is Hz, if the time scale is in milli-seconds and MHz, if the time scale is in nano-seconds, respectively.

In addition to the zooming arrows there exist an equivalent spin button which allows to move the starting point of the primary and filtered traces as well as of the corresponding spectra.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number** or by clicking in the primary file using the left mouse button.

The filter parameters for the individual processing steps are entered within the **filter parameter group**. The up/down arrows near the input fields allow to change the actual entered value by + or - 10 %. With the option **Apply on example trace** activated the effect of the filter parameters is online controlled when changing the parameters by showing both the original and filtered trace. The arrows near the input fields may be best used for such an interactive control. Deactivate this parameter if this control takes too much computer time.

After having specified the filter parameter you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing. After activating the processing label editing a list box appears which shows all labels already used for the current line.

Activating the option **sequence proc.** additionally open the Sequence Processing menu. With this menu opened you may easily add the current processing step to the sequence processing flow which may be applied to a choosable number of lines after.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing. Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing.

1.11.8.1 Running average

This filter acts on the chosen number of traces. The filter performs a running average over a selectable number of traces for each time step.

The running average is performed over a number of traces (parameter **average traces**). The max. bandwidth is restricted to 1024 traces to be averaged. For a bandwidth of 4 the current sample, the next two in horizontal direction to the left and the next two in horizontal direction to the right, i.e. five samples for each time value, are taken into account. If between the current sample and the first or the last trace are fewer samples than half the window width, the window width is decreased on one side. From these five samples the mean value is calculated and assigned to the current sample as new value. The filter is the more effective the larger the selected bandwidth.

It is possible to restrict the affected time area of the filter. For this you must enter within the filter parameter group the **start time** and the **end time** for which the filter shall be applied on. The resting data part will remain unfiltered. By default start time is set to the time-begin of each trace and end time is set to the max. time of each trace. The parameter **samples** within the taper width window allows to enter the number of samples for tapering the time start and end.

This filter method suppresses trace dependent noise. Its effect is to emphasize horizontally coherent energy.

1.11.8.2 Subtracting average

This filter acts on the chosen number of traces. The filter performs a subtracting average over a selectable number of traces for each time step. The filter performs a so called sliding background removal.

The subtracting average is performed over a number of traces (parameter **average traces**). The max. bandwidth is restricted to 1024 traces to be averaged. For a bandwidth of 4 the current sample, the next two in horizontal direction to the left and the next two in horizontal direction to the right, i.e. five samples for each time value, are taken into account. If between the current sample and the first or the last trace are fewer samples than half the window width, the window width is decreased on one side. From these five samples the mean value is calculated. This mean value is subtracted from the value of the current sample and the result is assigned to the current sample as new value. The filter is the more effective the smaller the selected bandwidth.

3 different averaging methods are supported. **Mean1** and **mean2** use two different methods defining a mean value (mean1 means arithmetic mean, mean 2 uses a special effect-attenuated algorithm), **median** uses a method defining the median value. In case of the median method an additional parameter **average samples** may be entered which allows to extend the average range in sample direction in addition.

It is possible to restrict the affected time area of the filter. For this you must enter within the filter parameter group the **start time** and the **end time** for which the filter shall be applied on. The resting data part will remain unfiltered. By default start time is set to the time-begin of each trace and end time is set to the max.

time of each trace. The parameter **samples** within the taper width window allows to enter the number of samples for tapering the time start and end.

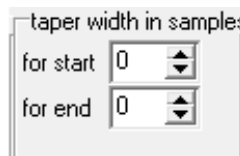
This filter method suppresses horizontally coherent energy. Its effect is to emphasize signals which vary laterally (e.g. diffractions).

1.11.8.3 Background removal

This filter acts on the chosen number of traces. The filter performs a subtracting of an averaged trace (tracrange) which is built up from the chosen time/distance range of the current section. The filter performs a so-called background removal.

For this you must define the time/distance range for building up the average trace. This average trace is shown in the reference trace window. The time range is defined by the filter parameters **start time** and **end time**. The **taper width in samples** option allows to enter the number of samples for tapering the time start and end.

The distance range is defined by the filter parameters **start distance** and **end distance**. It is also possible to define this data range interactively within the original profile. Click on the uppermost left corner of the area to be chosen and drag the mouse to the wanted lowermost right corner with the left mouse button pressed.



For each selected time sample the selected traces are averaged automatically in x-direction so that you get a time series for the subsequent subtraction. This time series is subtracted from every trace of the profile. The noise signal you wish to eliminate for example could be the range of the first arrival which is superposed locally with diffraction arrivals of shallow diffractors. Then you have to select a range of first arrivals where no diffractions are expected. Other noise that can be eliminated is, e.g. electronic signals.

With the definition of the time range for building up the time series the affected time area of the filter is automatically restricted to that range. The resting data part will remain unfiltered.

To be considered for the sequence processing: If the lines have different distance or time ranges it might be useful to select the start distance and end distance and/or the start time and end time in such a way that the largest ranges occurring within the lines are taken into account. Then it is sure that the filtertrace is built up for all traces for each line.

The radio groupbox allows you to specify whether the reference trace is determined from the **whole line**, from a special **distance range** (and **dist.range only** respectively) or whether it is automatically determined for **line parts** separated by each other by a new ensemble number. When using whole profile or line parts the filter parameters start distance and end distance are of no importance because always all traces within the whole profile or within one ensemble group are used for building up the filter trace. Activate the option line parts if the file is composed of different 2D-lines or shots (e.g. a 3D-file consisting of different 2D-parallel lines).

With the option **dist.range only** activated only the traces lying within the chosen distance range are filtered in contrast to the option distance range where only the

filter trace is determined from the chosen distance range but all traces are filtered.

If the option **external trace** has been selected the reference trace which will be subtracted from all traces of the profile will be loaded from an independent Reflexw datafile. The Reflexw datafiles may contain one or more traces but in any case only the first trace will be taken as the reference trace. The option load wavelets allows to reload the reference trace. The reference trace may contain a different timeincrement but different start times will not be taken into account.

With the option background removal you can eliminate temporally consistent noise from the whole profile and therefore possibly make signals visible, previously covered by this noise. This filter method also suppresses horizontally coherent energy. Its effect is also to emphasize signals which vary laterally (e.g. diffractions).

attention: it might happen that the filter causes non real signals. This holds true when the averaged time series contains energy which is not present within any part(s) of the profile.

1.11.8.4 Compress

This option allows both a compression in time-(Y-)direction and in distance-(X-)direction.

The filter parameters **factor (x-dir.)** and **factor (y-dir.)** define the size of the compressing window in x- and y-direction, this means every which trace and every which sample will be saved. The filter parameter start trace means, which trace will be used. The filter parameter start sample means which sample will be used. Example: factor (x-dir.): 3, factor (y-dir.): 2, start trace: 2, start sample: 1. The data are reduced to every 3. trace and 2. sample starting at trace number 2 and sample number 1. The samples or traces not needed anymore are omitted. Make sure before applying this option that due to compressing no aliasing problems occur.

The radio box **compress** controls the handling of new traceincrement. With the option **fix profile length** activated the new traceincrement will be changed according to the entered factor (x-dir.). With the option **fix traceincrement** activated the new traceincrement will not be changed resulting in a smaller profile length according the entered factor (x-dir.).

A compression is reasonable before the application of computing-time intensive processing steps (e.g. fk-Filter).

1.11.8.5 Expand

This option allows both an expansion in time-(Y-)direction and in distance-(X-)direction.

The filter parameters **factor (x-dir.)** and **factor (y-dir.)** define the size of the expansion window in x- and y-direction. The values between the old sample values or traces, respectively, are linearly interpolated.

With option **keep 0 value(0,1)** set to 1 the interpolation will be restricted to non 0 values. This might be useful for example for a tomographic result.

Please note that with the application of this option the resolution capacity is not increased. Though, a reasonable application is for example an enhanced visualization.

1.11.8.6 Stack traces

The option allows the temporal simultaneous stacking of a selectable number of traces.

As filter parameter enter the **nr. of traces** to be stacked. In contrast to the option compress here no data are omitted but the mean value for the chosen number of traces is calculated for each time sample. The total number of traces decreases according to the number of stack traces. Please make sure before applying this option that no aliasing problems occur due to stacking. The traceincrement will be changed accordingly - see below.

With the parameter **nr.of tr./ensemble** greater 1 the stacking may not be done on successive traces but for common successive ensembles. Example: A stacking shall be done for shot ensembles which have been acquired four times at the same position. Each shot ensemble includes 48 traces. Then you have to enter nr. of traces 4 and nr.of tr./ensemble 48. The result are stacked shot ensembles stored sequentially. Another example: 3 constant offset profiles with 1000 traces each stored sequentially within one file. Enter 3 for nr.of traces and 1000 for nr. tr./ensemble. If nr.of tr./ensemble is greater than 1 the traceincrement will not be changed.

With the parameter **nr.of tr./ensemble** smaller/equal 1 (default) the stacking is based on successive traces.

The filter might be applied in order to enhance the signal to noise ratio.

1.11.8.7 Subtract traces

The option allows the temporal simultaneous subtraction of a selectable number of traces from the first trace within each group.

As filter parameter enter the **nr. of traces** to be subtracted from the first trace within each group. The number of traces within each group is nr. of traces + 1. A useful parameter value is 1. In this case always the second trace is subtracted from the first one within a group of 2 traces.

The filter might be applied for shear wave-data with different polarities of the p-waves. The data must be sorted in such a way that subsequent traces have the different polarities.

1.11.8.8 Compress 3D-file

This option allows a compression of a 3D-file into the two directions. A 3D-file consists of several parallel 2D-lines separated by the so called line increment.

The filter parameters **factor (line dir.)** and **factor (line perp.)** define the size of the compressing window in line-direction and perpendicular to the line direction. The filter parameter **start trace** means, which trace will be used first. The filter parameter **start 2D-line** means which 2D-line will be used first. Example: factor (line dir.): 3, factor (line perp.): 2, start trace: 2, start 2D-line: 1. The data are reduced to every 3. trace in line direction and 2. trace perpendicular to the 2D-lines starting at trace number 2 and 2D-line number 1. The samples or traces not needed anymore are omitted. Make sure before applying this option that due to compressing no aliasing problems occur.

A compression is reasonable before the application of computing-time intensive processing steps (e.g. 3D-migration, see chap. 1.11.9).

1.11.8.9 Expand 3D-file

This option allows to expand a 3D-file perpendicular to the profile direction. A 3D-file consists of several parallel 2D-lines separated by the so called line increment.

The filter parameters **factor (line dir.)** and **factor (line perp.)** define the size of the expanding window in line-direction and perpendicular to the line direction. The number of 2D-lines building up the 3D-file changes accordingly and also the traceincrement and/or lineincrement will be changed according to the entered parameter factors. The start and end coordinates will not be changed. Therefore the number of 2D-lines only changes to $((\text{old number per line}-1) * (\text{factor inline}) + 1) * ((\text{old ensemble number}-1) * (\text{factor perp}) + 1)$.

1.11.8.10 average xy-filter

The filter calculates the average over a selectable xy-area for each time step.

The average is performed both over a number of traces (parameter **nr. of traces (x)**) and over a number of samples (parameter **nr. of samples (y)**). The max. bandwidth in x-direction is restricted to 256 traces to be averaged. The filter area is centered around the current data point. Example: nr of traces (x) is set to 4 and number of samples (y) is set to 8: around the current data point a data area of 5*9 data points used for determine the average. The filter is the more effective the larger the selected area.

It is possible to restrict the affected time area of the filter. For this you must enter within the filter parameter group the **start time** and the **end time** for which the filter shall be applied on. The resting data part will remain unfiltered. By default start time is set to the time-begin of each trace and end time is set to the max. time of each trace.

This filter method suppresses trace and time dependent noise. It acts as a lowpass filter both in x- and y-(time) direction.

1.11.8.11 median xy-filter

The filter calculates the median over a selectable xy-area for each time step.

The median is performed both over a number of traces (parameter **nr. of traces (x)**) and over a number of samples (parameter **nr. of samples (y)**). The max. bandwidth in x-direction is restricted to 256 traces to be averaged. The filter area is centered around the current data point. Example: nr of traces (x) is set to 4 and number of samples (y) is set to 8: around the current data point a data area of 5*9 data points used for determine the median. The filter is the more effective the larger the selected area.

It is possible to restrict the affected time area of the filter. For this you must enter within the filter parameter group the **start time** and the **end time** for which the filter shall be applied on. The resting data part will remain unfiltered. By default start time is set to the time-begin of each trace and end time is set to the max. time of each trace.

This filter method suppresses trace and time dependent noise and spike events. It acts as a lowpass filter both in x- and y-(time) direction.

1.11.8.12 dilation xy-filter

Dilation and erosion are morphology-based operations. Dilation, in general, causes objects to dilate or grow in size but also a hole fitting will be observed.

The **filter size** must be entered. 3*3 disk/li. and 3*3 disk includes 5 points, 3*3 square 9 points points, 5*5 disk 13 points, 5*5 square 25 points and 7*7 disk 29 points. The filter effect increases with the filtersize (3*3 disk/lin. has the smallest effect). The filter area is centered around the current data point.

It is possible to restrict the affected time (y-) area of the filter. For this you must enter within the filter parameter group the **start time** and the **end time** for which the filter shall be applied on. The resting data part will remain unfiltered. By default start time is set to the time-begin of each trace and end time is set to the max. time of each trace.

The filter is mainly useful for the display of slices.

1.11.8.13 erosion xy-filter

Erosion causes objects to shrink. The combination with the dilation xy-filter has a similar effect like the median xy filter.

The **filter size** must be entered. 3*3 disk/li. and 3*3 disk includes 5 points, 3*3 square 9 points, 5*5 disk 13 points, 5*5 square 25 points and 7*7 disk 29 points.

The filter effect increases with the filtersize (3*3 disk/lin. has the smallest effect). The filter area is centered around the current data point. It is possible to restrict the affected time (y-) area of the filter. For this you must enter within the filter parameter group the **start time** and the **end time** for which the filter shall be applied on. The resting data part will remain unfiltered. By default start time is set to the time-begin of each trace and end time is set to the max. time of each trace.

The filter is mainly useful for the display of slices.

1.11.8.14 constrain borders

The option checks the non convered range and sharpens the borders depending on the entered parameters. The option should only be applied on timeslices.



The parameters **nr. of traces (x)** and **nr. of samples (y)** define the range for sharpening and may for example coincide with the interpolation range used for the calculation of the timeslices (see below).

The option might be useful if an interpolation has been used for generating the timeslices and due to this interpolation the borders of the acquisition area have been enlarged and smoothed. This might be critical if the acquisition area has some obstacles like walls and the interpolation ranges into these (picture on the left). The option now allows you to sharpen the acquisition borders and to remove the extrapolation into non desired ranges (picture on the right).

1.11.8.15 average 3D-filter

The filter acts on a 3D-file and performs an averaging in all 3 directions. The option **nr. of voxels in time** controls the number of voxels in z(time)-direction for determining the average, **nr. of voxels inline** controls the number of voxels in inline direction and **nr. of voxels crossline** controls the number of voxels in crossline direction.

1.11.9 Migration

The following migration and timedepth conversion possibilities are included in the migration/time-depth conversion group:

- diffraction stack
- Kirchhoff migration
- fk migration (Stolt)
- topography migration
- diffraction 2D-veloc
- Kirchhoff 2D-veloc
- FD-migration
- timedepth conversion
- semblance
- prestack migration
- 3D diffraction stack
- 3D Kirchh. migration
- 3D-fk migration
- 3D-Kirchh. 2D-vel.
- 3D-semblance

The arrows near the original trace and original spectrum are used to enhance the resolution of the trace and spectrum windows. The trace or the spectrum is stretched up and down when using these arrows. The spectrum scale is Hz, if the time scale is in milli-seconds and MHz, if the time scale is in nano-seconds, respectively.

In addition to the zooming arrows there exist an equivalent spin button which allows to move the starting point of the primary and filtered traces as well as of the corresponding spectra.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number** or by clicking in the primary file using the left mouse button.

The filter parameters for the individual processing steps are entered within the **filter parameter group**. The up/down arrows near the input fields allow to change the actual entered value by + or - 10 %. With the option **Apply on example trace** activated the effect of the filter parameters is online controlled when changing the parameters by showing both the original and filtered trace. The arrows near the input fields may be best used for such an interactive control. Deactivate this parameter if this control takes too much computer time.

After having specified the filter parameter you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing. After activating the processing label editing a list box appears which shows all labels already used for the current line.

Activating the option **sequence proc.** additionally open the Sequence Processing menu. With this menu opened you may easily add the current processing step to the sequence processing flow which may be applied to a selectable number of lines after.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing. Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing.

1.11.9.1 Diffraction stack

A simple time migration (diffraction stack) of a two-dimensional profile on the basis of a constant velocity is performed. The profile must represent a so called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source".

The **diffraction stack** is done in the x-t range, this means that an unweighted summation for each point of the profile over a calculated hyperbola of preset bandwidth is performed. The bandwidth means the number of traces (parameter **summation width**) over which shall be summed. Here the total number of traces (on both sides) has to be specified. A larger number of traces naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbola that extend over a large trace range. With a summation width smaller than 256 the calculation of the hyperbola is only done once which also saves a lot of computation time.

Additionally the **velocity** has to be specified in the dimension predefined by the distance and time dimension. An overview over the optimal velocity can be obtained within the option velocity adaptation where the calculated hyperbola can be optimally adapted to existing diffraction hyperbola by interactive continuous variation of the velocity. The precondition for good determination of the velocity is the existence of diffraction arrivals. Here, also the success of the migration can be realized best. Ideally they should shrink to a very small area. Beside the contraction of the diffraction energy another aim of migration is to shift the arrivals to their "true" position. This is important for steep dipping reflectors. The time range for applying the migration may be restricted. The filter parameters **start time** and **end time** define the time range.

With the option **adapt width** within the migration box active the summation width will be automatically adapted to the depth. The parameter **max. angle** (theta in degrees) controls the max. angle for the summation. As a result the summation width becomes time-dependent (linear increase with time) and reaches the entered max. summation width at that time which is defined by the entered max. angle, the summation width and the velocity. This base time is shown at right of the entered angle and will be updated if any of these parameters will be changed. Activating the option adapt width allows a larger max. summation width and may be useful if the real diffractions exhibit a smaller width at smaller times due to the source and receiver characteristics.

The switch groupbox allows you to specify whether the summation is performed within the **whole profile** or whether individual data parts are automatically determined for **line parts** separated by each other by a new ensemble number. When using line parts, the migration is restricted to the area of each individual linepart. Activate the option line parts, if the file is composed of different 2D-lines or shots (e.g. a 3D-file consisting of different 2D-parallel lines) because otherwise data are summed up which have not been acquired within one 2D-line.

The migration is one of the most important filters. A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions

are present, the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

1.11.9.2 Kirchhoff migration

A time migration of a two-dimensional profile on the basis of a constant velocity is performed. The profile must represent a so-called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source".

The **Kirchhoff migration** is done in the x-t range, this means that a weighted summation for each point of the profile over a calculated hyperbola of preset bandwidth is performed. Before the summation the original profile is differentiated. The bandwidth means the number of traces (parameter **summation width**) over which shall be summed. Here the total number of traces (on both sides) has to be specified. A larger number of traces naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbola that extend over a large trace range. With a summation width smaller than 256 the calculation of the hyperbola is only done once which also saves a lot of computation time.

Additionally the **velocity** has to be specified in the dimension predefined by the distance and time dimension. An overview over the optimal velocity can be obtained within the option velocity adaptation where the calculated hyperbola can be optimally adapted to existing diffraction hyperbola by interactive continuous variation of the velocity. The precondition for good determination of the velocity is the existence of diffraction arrivals. Here, also the success of the migration can be realized best. Ideally they should shrink to a very small area. Beside the contraction of the diffraction energy another aim of migration is to shift the arrivals to their "true" position. This is important for steep dipping reflectors. The time range for applying the migration may be restricted. The filter parameters **start time** and **end time** define the time range.

With the option **adapt width** within the migration box active the summation width will be automatically adapted to the depth. The parameter **max. angle** (theta in degrees) controls the max. angle for the summation. As a result the summation width becomes time-dependent (linear increase with time) and reaches the entered max. summation width at that time which is defined by the entered max. angle, the summation width and the velocity. This base time is shown at right of the entered angle and will be updated if any of these parameters will be changed. Activating the option adapt width allows a larger max. summation width and may be useful if the real diffractions exhibit a smaller width at smaller times due to the source and receiver characteristics.

The switch groupbox allows you to specify whether the summation is performed within the **whole profile** or whether individual data parts are automatically determined for **line parts** separated by each other by a new ensemble number. When using line parts, the migration is restricted to the area of each individual linepart. Activate the option line parts, if the file is composed of different 2D-lines or shots (e.g. a 3D-file consisting of different 2D-parallel lines) because otherwise data are summed up which have not been acquired within one 2D-line.

The migration is one of the most important filters. A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions

are present, the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

1.11.9.3 Fk migration (Stolt)

A fast **fk-migration** of a two-dimensional profile after Stolt (1978, Geophysics 43, 23-48) on the basis of a constant velocity is performed. The profile must represent a so-called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source".

The method works in the frequency-wavenumber (fk) range (see also fk spectrum). Within the fk-range a variable transform is done based on the entered constant **velocity** (frequency is transformed onto the vertical wavenumber). If only a small data volume in time direction is available one should use an interpolation for this transformation. Such an interpolation is not very stable. That is why such an interpolation is not used in the program. This means the data volume in the f-k-range in direction of the variable to be transformed (in this case the frequency) must be large enough. That is why the Stolt-migration should only be used if the number of wavelengths inside the time window to be considered exceeds a critical value (about 10-15). If this does not hold true, the time window must be enlarged. This is done if the filter **scaling factor** is set to a value larger than 1. This factor allows to automatically exceed the time range by the given value (e.g. enter a factor 2: the time range is doubled and therefore the number of wavelengths within this window).

The time range for applying the migration may be restricted. The filter parameters **start time** and **end time** define the time range.

The switch groupbox allows you to specify whether the migration is performed within the **whole profile** or whether individual data parts are automatically determined for **line parts** separated by each other by a new ensemble number. When using line parts, the migration is restricted to the area of each individual linepart. Activate the option line parts if the file is composed of different 2D-lines or shots (e.g. a 3D-file consisting of different 2D-parallel lines) because otherwise data are summed up which have not been acquired within one 2D-line.

First the x-t data are transformed into the f-k-range (see also option fk-filter). After having done the transformation the migration process will be done and after that the back transformation into the x-t-range is performed.

In comparison to the Kirchhoff migration one has not to enter a summation width but the "summation" is always done using all traces. Therefore the Kirchhoff migration often gives better results if a restriction of the summation width is desired (e.g. if the diffractions only extend over a few traces). The Stolt migration is mostly preferable for the migration of extended or steep reflectors.

The migration is one of the most important filters. A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions are present, the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

1.11.9.4 topography migration

A time migration of a two-dimensional profile on the basis of a constant velocity under consideration of a surface topography is performed. The profile must represent a so-called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source". The migration automatically takes into account the different raypaths when a strong surface topography is present.

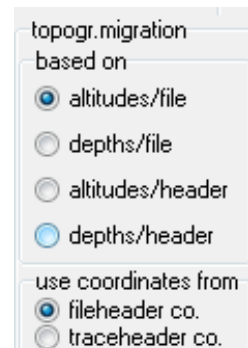
The **topography migration** is done in the x-t range, this means that a weighted summation for each point of the profile over a calculated hyperbola of preset bandwidth is performed (see also Kirchhoff migration). Before the summation the original profile is differentiated. The bandwidth means the number of traces (parameter **summation width**) over which shall be summed. Here the total number of traces (on both sides) has to be specified. A larger number of traces naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbola that extend over a large trace range.

Additionally the **velocity** has to be specified in the dimension predefined by the distance and time dimension. An overview over the optimal velocity can be obtained within the option velocity adaptation where the calculated hyperbola can be optimally adapted to existing diffraction hyperbola by interactive continuous variation of the velocity. The precondition for good determination of the velocity is the existence of diffraction arrivals. Here, also the success of the migration can be realized best. Ideally they should shrink to a very small area. Beside the contraction of the diffraction energy another aim of migration is to shift the arrivals to their "true" position. This is important for steep dipping reflectors. The time range for applying the migration may be restricted. The filter parameters **start time** and **end time** define the time range.

The switch groupbox allows you to specify whether the summation is performed within the **whole profile** or whether individual data parts are automatically determined for **line parts** separated by each other by a new ensemble number. When using line parts, the migration is restricted to the area of each individual linepart. Activate the option line parts, if the file is composed of different 2D-lines or shots (e.g. a 3D-file consisting of different 2D-parallel lines) because otherwise data are summed up which have not been acquired within one 2D-line.

There are two different ways for specifying where the **surface topography** will be taken from. The topographic values are loaded either from an external ASCII-file (options altitudes/file and depths/file) or from the values given within the traceheaders (options altitudes/header and depths/header, receiver z-coordinate). If the topographic value are taken from an external ASCII-file you can choose where the original profile coordinates for the synchronization shall be taken from - either from the overall fileheader or from the individual traceheader coordinates.

The format of the ASCII-file corresponds to the format used for the option Correct 3D-topography (see chap.



1.11.3.11) with each line containing one set one set of 3D coordinates:

x-position y-position z-position

The x- and y-position define the acquisition plane where the profiles are located.

The z-position defines the topographic niveau (either altitudes or depths). At least one blank must exist between the individual coordinates. A linear interpolation will be done between the given topographic values. Example with 3 different coordinate sets:

0 0 10

1 0 9

2 0 9.5

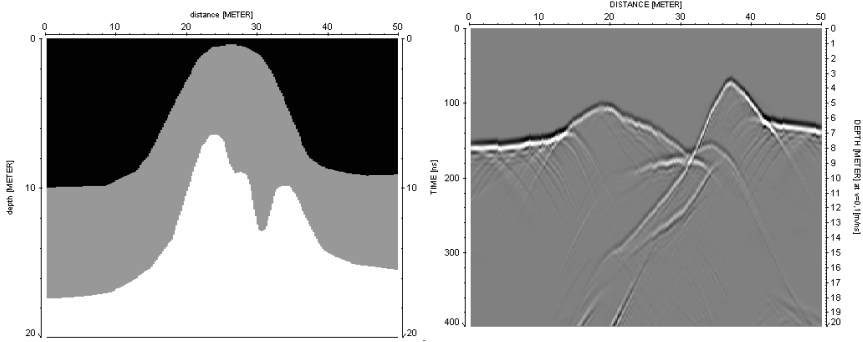
To be considered: the coordinates of the ASCII-file may either define the true x- and y-positions or they have been determined directly on the topographic interface. In both cases the original profile coordinates must define the same level.

It is possible to restrict the summation width using the option **adapt width**. The parameter **max. angle** (theta in degrees) controls the max. angle for the summation. As a result the summation width becomes time-dependent (linear increase with time). Activating the option **adapt** may be useful if the real diffractions exhibit a smaller width due to the source and receiver characteristics.

A topographic correction is not done automatically. The resulting migrated section still represents a section with the receivers located on the surface. If a topographic correction is wanted the processing step **Correct 3D-topography** must still be performed in addition (see also the following synthetic example). To be considered: the topographic correction must always be performed after the topography migration.

The topography migration should be used if a strong topography is present with undulations which are comparable to the depths of the target structures. A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions are present, the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

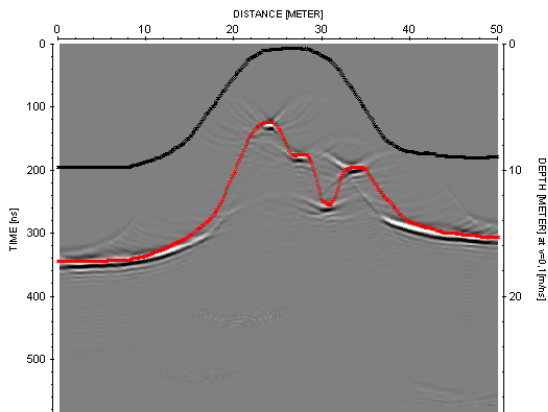
The following pictures show a synthetic data example with a strong surface topography (source and receiver are located on this surface topography). The depths of the target reflector are comparable to the undulations of the surface relief. The model is shown on the lower left panel and the simulated zero-offset with the receivers and the sources located on the surface topography is shown on the right side (raw data - no topographic correction has been applied).



Three different processing sequences have been performed which are shown on the next page.

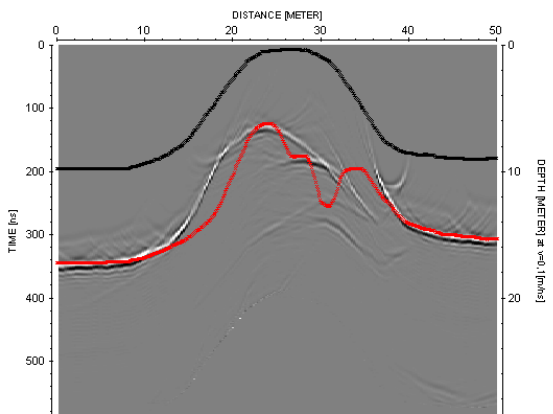
1. Correct 3D-topography
2. Kirchhoff migration with $v=0.1$ m/ns and summation over 100 traces

The reconstruction of the reflector is not perfect.



1. Kirchhoff migration with $v=0.1$ m/ns and summation over 100 traces
2. Correct 3D-topography

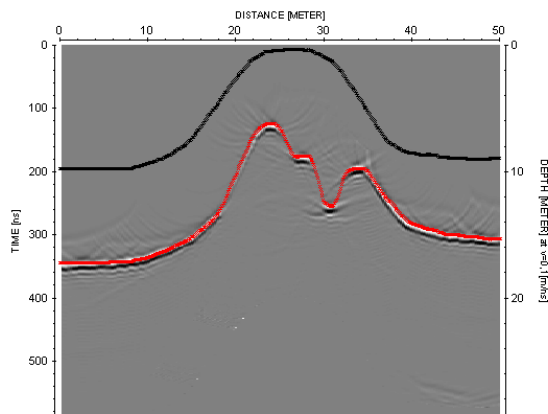
The reconstruction of the reflector is poor.



1. topography migration with $v=0.1$ m/ns and summation over 100 traces
2. Correct 3D-topography

The reconstruction is nearly perfect.

The best result is given with the topography migration followed by the option Correct 3D-topography.



1.11.9.5 diffraction 2D-veloc

A simple time migration (diffraction stack) of a two-dimensional profile on the basis of a 2D-velocity distribution is performed. The profile must represent a so-called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source".

The **diffraction stack** is done in the x-t range, this means that an unweighted summation for each point of the profile over a calculated hyperbola of preset bandwidth is performed. The bandwidth means the number of traces (parameter **summation width**) over which shall be summed. Here the total number of traces (on both sides) has to be specified. A larger number of traces naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbola that extend over a large trace range. With a summation width smaller than 128 the calculation of the hyperbola is only done once which also saves a lot of computation time.

The time range for applying the migration may be restricted. The filter parameters **start time** and **end time** define the time range.

The radio groupbox allows you to specify whether the 2D-velocity distribution is obtained from the **CMP-analysis** (see also CMP 2D-model) or from the **hyperbola adaptation** (see also velocity adaptation) or directly read from a Reflexw formatted **rasterfile**. If the velocity adaptation had been only done using cores it is recommended to use the CMP-analysis button because in this case the saved 1D-velocity files will be used for the 2D-velocity distribution.

The **rasterfile** may either contain the mean velocities (option **rasterfile mean**) or the layervelocities (option **rasterfile layer**). The velocities (amplitudes) must be in m/ns or epsilon for GPR data and in m/s for seismic or acoustic data. The rasterfile may either be a depth model (y-axis always meter) or a twoway travelttime model (y-axis corresponds to the two way travelttime). The x-axis is always the distance.

The current VRMS- or average velocity for each point of the profile is determined from the 2D-distribution and the summation is performed with this velocity, i.e. for the calculation of the current diffraction hyperbola only a constant velocity is assumed. However this is a valid approximation if the diffraction branches do not extend over a large laterally heavily fluctuating area. For Ground Penetrating Radar data this is normally always guaranteed, for seismic data the summation width has to be reduced accordingly in the case of strong lateral velocity fluctuations. If the migration shall be done using the "correct" 2D-velocity distribution, the FD-migration must be used.

After having started the migration, the desired 2D-velocity distribution going to be the basis of the migration is queried. The 2D-velocity distribution is shown in a separate window.

The switch groupbox data subdivision allows you to specify whether the summation is performed within the **whole profile** or whether individual data parts are automatically determined for **line parts** separated by each other by a new ensemble number. When using line parts, the migration is restricted to the area of each individual linepart. Activate the option line parts, if the file is composed of different 2D-lines or shots (e.g. a 3D-file consisting of different 2D-parallel lines)

because otherwise data are summed up which have not been acquired within one 2D-line.

The migration is one of the most important filters. A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions are present, the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

1.11.9.6 Kirchhoff 2D-veloc

A time migration of a two-dimensional profile on the basis of a 2D-velocity distribution is performed. The profile must represent a so-called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source".

The **Kirchhoff 2D-veloc** migration is done in the x-t range, this means that a weighted summation for each point of the profile over a calculated hyperbola of preset bandwidth is performed. The bandwidth means the number of traces (parameter **summation width**) over which shall be summed. Here the total number of traces (on both sides) has to be specified. A larger number of traces naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbola that extend over a large trace range. With a summation width smaller than 128 the calculation of the hyperbola is only done once which also saves a lot of computation time.

The time range for applying the migration may be restricted. The filter parameters **start time** and **end time** define the time range.

The radio groupbox allows you to specify whether the 2D-velocity distribution is obtained from the **CMP-analysis** (see also CMP 2D-model) or from the **hyperbola adaptation** (see also velocity adaptation) or directly read from a Reflexw formatted **rasterfile**. If the velocity adaptation had been only done using cores it is recommended to use the CMP-analysis button because in this case the saved 1D-velocity files will be used for the 2D-velocity distribution.

The **rasterfile** may either contain the mean velocities (option **rasterfile mean**) or the layer velocities (option **rasterfile layer**). The velocities (amplitudes) must be in m/ns or epsilon for GPR data and in m/s for seismic or acoustic data. The rasterfile may either be a depth model (y-axis always meter) or a twoway traveltime model (y-axis corresponds to the two way traveltime). The x-axis is always the distance.

The current VRMS- or average velocity for each point of the profile is determined from the 2D-distribution and the summation is performed with this velocity, i.e. for the calculation of the current diffraction hyperbola only a constant velocity is assumed. However this is a valid approximation if the diffraction branches do not extend over a large laterally heavily fluctuating area. For Ground Penetrating Radar data this is normally always guaranteed, for seismic data the summation width has to be reduced accordingly in the case of strong lateral velocity fluctuations. If the migration shall be done using the "correct" 2D-velocity distribution, the FD-migration must be used.

After having started the migration, the desired 2D-velocity distribution going to be the basis of the migration is queried. The 2D-velocity distribution is shown in a separate window.

The switch groupbox data subdivision allows you to specify whether the summation is performed within the **whole profile** or whether individual data parts are automatically determined for **line parts** separated by each other by a new ensemble number. When using line parts, the migration is restricted to the area of each individual linepart. Activate the option line parts, if the file is composed of different 2D-lines or shots (e.g. a 3D-file consisting of different 2D-parallel lines) because otherwise data are summed up which have not been acquired within one 2D-line.

The migration is one of the most important filters. A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions are present, the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

1.11.9.7 FD-migration

A time FiniteDifference(FD) migration of a two-dimensional profile on the basis of a 2D-velocity distribution is performed. The profile must represent a so-called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source".

The **FD-migration** is done based on the one-way scalar wave equation. A 1st approximation is used.

The time range for applying the migration may be restricted. The filter parameters **start time** and **end time** define the time range. The parameter **nom.frequency** defines the main frequency of the signal in GHz (timedimension=ns) or Hz (timedimension ms or μ s), respectively. These parameters are also stored within the fileheader.

The radio groupbox allows you to specify whether the 2D-velocity distribution is obtained from the **CMP-analysis** (see also CMP 2D-model), from the **hyperbola adaptation** (see also velocity adaptation) or directly read from a Reflexw formatted **rasterfile**. The rasterfile must contain the layer velocities (option **rasterfile layer**). The velocities (amplitudes) must be in m/ns or epsilon for GPR data and in m/s for seismic or acoustic data. The rasterfile must be a depth model with the dimensions in x and z identical to the distance dimension of the original line.

If hyperbola adaptation has been chosen it must be considered that the FD-migration needs a layered medium. Therefore the hyperbola adaptation velocities are also interpreted as layer velocities. If the velocity adaptation had been only done using cores it is recommended to use the CMP-analysis button because in this case the saved 1D-velocity files will be used for the 2D-velocity distribution.

In contradiction to the Kirchhoff 2D-veloc and diffraction 2D-veloc migration methods the "correct" 2D-velocity distribution is used for the downward continuation of the wavefield.

After having started the migration, the desired 2D-velocity distribution going to be the basis of the migration is queried. The 2D-velocity distribution is shown in a separate window.

The migration is one of the most important filters. A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions are present, the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

1.11.9.8 timedepth conversion

This option allows the conversion of the zero-offset-section into a depth-section. This is not (!) a depth migration but a transformation of the time axis into a depth axis on the basis of a certain 2D-velocity distribution or on a constant velocity.

The timedepth conversion may be done based on a **2D-velocity** or on a **const.velocity**.

If **const. velocity** has been chosen only the **velocity** must be entered manually.

If **2D-velocity** has been chosen the following parameters must be entered:

The filter parameters **min.depth** and **max.depth** define the new y-axis range. The parameters are given in the distance dimension of the x-axis.

The radio groupbox allows you to specify whether the 2D-velocity distribution is obtained from the **CMP-analysis** (see also CMP 2D-model) or from the **hyperbola adaptation** (see also velocity adaptation) or directly read from a Reflexw formatted **rasterfile**. If the velocity adaptation had been only done using cores it is recommended to use the CMP-analysis button because in this case the saved 1D-velocity files will be used for the 2D-velocity distribution.

The **rasterfile** may either contain the mean velocities (option **rasterfile mean**) or the layervelocities (option **rasterfile layer**). The velocities (amplitudes) must be in m/ns or epsilon for GPR data and in m/s for seismic or acoustic data. The rasterfile may either be a depth model (y-axis always meter) or a twoway travelttime model (y-axis corresponds to the two way travelttime). The x-axis is always the distance.

After having started the timedepth conversion, the desired 2D-velocity distribution going to be the basis of the timedepth conversion is queried. The 2D-velocity distribution is shown in a separate window.

To be considered: because of the possible changes of the mean velocities both in x- and y-direction the data and therefore the shape of the signals may be drastically changed. This also may result in great changes of the apparent "frequency" of the signals.

1.11.9.9 semblance

This option allows to calculate the semblance or the unnormalized correlation for each datapoint within the zero-offset section based on a summation over a calculated hyperbola.

The **semblance analysis** is done in the x-t range, this means that an unweighted summation for each point of the profile over a calculated hyperbola of preset bandwidth is performed. The bandwidth means the area in x-direction for summation in the given distance dimension (parameter **width**) over which shall be summed. A larger area naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbola that extend over a large trace range.

Additionally the **velocity** has to be specified in the dimension predefined by the distance and time dimension. An overview over the optimal velocity can be obtained within the option velocity adaptation where the calculated hyperboloid can be optimally adapted to existing diffraction hyperbola by interactive continuous variation of the velocity. Precondition for good determination of the velocity is the existence of diffraction arrivals.

In addition you must specify the **nom.frequency** in the given frequency dimension. This parameter is necessary if the envelope (parameter **envelope** activated within the semblance groupbox) of the semblance shall be calculated. If the nominal frequency is stored within the file header (see also chap. 1.4.3) this value is taken as the default value.

Within the semblance groupbox you also may specify if the semblance data shall be normalized to 1 or to the max. value of the original profile.

1.11.9.10 prestack migration

The prestack migration allows to trace back the wavefield of single shots or constant offset data to their "source". The migration takes into account arbitrary shot/receiver offsets in contrast to the other migration methods described above which need zero offset data as input. The following geometries are supported:

- shots and receivers are located along the x-axis including a topography ("normal" **reflection mode**)
- shots or receivers are located along the x-axis and shots or receivers are located along the y(z)-axis (borehole) - walk away vertical seismic profile (**WVSP**)

- shots and receivers are both located along the y(z) axis. The x-postion may be different but fixed for the shots and the receivers- **crosshole**

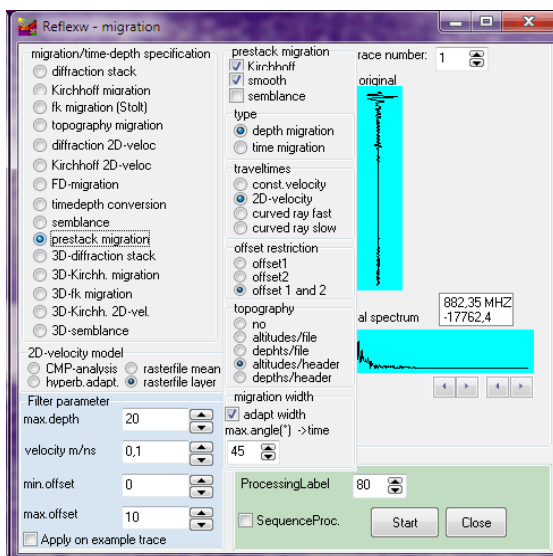
The program automatically controls the correct geometry.

In the second and third case the curvers ray methods (see below) must be used. The prestack migration calculates traveltimes curves for each transmitter-receiver pair on the base of a constant velocity or of a 2D-velocity field. Based on these traveltimes a summation for each underground point (x,z) is performed. The result is a zero offset (ZO) depth migrated section.

If applied on **single shots** or other **multioffset data** the processing step prestack migration replaces the complete "normal" processing flow without the need of any sorting and stacking (CMP-sorting -> stacking -> ZO migration -> timedepth conversion). The shot-x and receiver-x traceheadercoordinates are used for the calculation. The y-traceheadercoordinates are not taken into account.

In case of a **constant offset dataset** the overall fileheader coordinates (profilestart, traceincrement and S/R-distance) will be used for the prestack migration. In case of a S/R-distance $\neq 0$ it is assumed that the midpoint between source and receiver line has been defined as profilestart within the fileheader. Possible existing traceheader coordinates (e.g. GPS-coordinates) within the original data will be kept but these coordinates will not be used for the migration process.

The parameter **max. depth** defines the maximum depth range for the depth migration.



The parameter **velocity** defines the velocity in case of const.velocity migration (see below) and will be used for the smooth option if a depth migration will be performed.

The parameters **min. and max. offset** restrict the range for the summation (see below option offset restriction).

Several methods are supported:

const.velocity: the base for the calculation of the traveltimes curves is a constant velocity. This suboption can only be used if a quite homogeneous velocity distribution is given or for a first overview.

2D-velocity: the base for the traveltimes calculation is a 2D-velocity field. The current VRMS- or average velocity for each underground point (x,z) is determined from the 2D-distribution and the traveltimes are calculated using this velocity. This is only a valid approximation if the summation branches do not extend over a large laterally heavily fluctuating area.

curved ray slow and **curved ray fast**: the base for the traveltimes calculation is a 2D-velocity field. The traveltimes for each underground point are calculated using a Finite Difference approximation of the Eikonal equation (Vidale, 1988, see also modelling module) which has been proved to be very exact even for extremely complicated media. In the case of the option curved ray slow the number of different single calculations is equal the number of receivers. In the case of activated option curved ray fast the number equals the number of receivers multiplied with the number of gridpoint in z-direction and the calculation is therefore much slower. For most cases the accuracy using the option curved ray slow is high enough. Only if very strong velocity contrasts are given within the used 2D-velocity field the option curved ray slow may be a useful alternative.

The radio groupbox **2D-velocity model** allows you to specify whether the 2D-velocity distribution is obtained from the **CMP-analysis** (see also CMP 2D-model) or from the **hyperbola adaptation** (see also velocity adaptation) or directly read from a Reflexw formatted **rasterfile**. If the velocity adaptation had been only done using cores it is recommended to use the CMP-analysis button because in this case the saved 1D-velocity files will be used for the 2D-velocity distribution. The rasterfile may either contain the mean velocities (option **rasterfile mean**) or the layervelocities (option **rasterfile layer**). The velocities (amplitudes) must be in m/ns or epsilon for GPR data and in m/s for seismic or acoustic data. The rasterfile may either be a depth model (y-axis always meter) or a twoway traveltimes model (y-axis corresponds to the two way traveltimes). The x-axis is always the distance.

The seismic or GPR data can be expressed as a convolution of the impulse response of the underground and the source signal. The calculated traveltimes

prestack migration

☒ Kirchhoff

☒ smooth

☐ semblance

type

☒ depth migration

☐ time migration

traveltimes

☒ const.velocity

☐ 2D-velocity

☐ curved ray fast

☐ curved ray slow

offset restriction

☐ offset1

☐ offset2

☒ offset 1 and 2

topography

☐ no

☐ altitudes/file

☐ depths/file

☒ altitudes/header

☐ depths/header

migration width

☒ adapt width

max.angle(*) ->time

25

will be determined on the base of the impulse response of the underground. Therefore the summation is only correct for the first arrival. With activated option **smooth** the summation is done over a depth range corresponding to the real wavelength. This significantly may increase the data focussing. For a depth migration the overall velocity will be used for the calculation of the depth range. With activated option **Kirchhoff** weighted factors are used for the summation. The option **semblance** allows to calculate the semblance along the summations paths and multiplies the resulting semblance section with the migrated section. The option might be useful to enhance low amplitude reflections which exhibit a high semblance.

There are two possibilities to restrict the **offset range** for the summation. The min. and max. offsets are given in the distance dimension. Offset 1 means the min. and max. distance between the real transmitter- and receiver coordinates. Offset 2 means the min. and max. distance between underground point and shot- and receiver position.

The migration is a **depth migration** - therefore the result is a depth-distance section. Optionally the depth section may be transformed into two way traveltimes for the case of the constant and 2D-velocity methods (radio box type set to **time migration**).

There are different possibilities to include the **topography** directly into the migration process. The topographic values are loaded either from an external ASCII-file (options altitudes/file and depths/file) or from the values given within the traceheaders (options altitudes/header and depths/header). The format of the ASCII-file corresponds to the format used for the option Correct 3D-topography (see chap. 1.11.3.11) with each line containing one set one set of 3D coordinates:

x-position y-position z-position

The x- and y-position define the acquisition plane where the profiles are located. The z-position defines the topographic niveau (either altitudes or depths). At least one blank must exist between the individual coordinates. A linear interpolation will be done between the given topographic values. Example with 3 different coordinate sets:

```
0 0 10
1 0 9
2 0 9.5
```

A topographic correction is not done automatically. The resulting migrated section still represents a section with the receivers located on the surface. If a topographic correction is wanted the processing step Correct 3D-topography must still be performed in addition. To be considered: the topographic correction must always be performed after the prestack migration.

It is possible to restrict the summation width using the option **adapt width**. The parameter **max. angle** (theta in degrees) controls the max. angle for the summation. As a result the summation width becomes timedependent (linear increase with time). Activating the option adapt may be useful if the real diffractions exhibit a smaller width due to the source and receiver characteristics

1.11.9.11 3D-diffraction stack

A simple time 3D-migration (diffraction stack) of a three-dimensional profile on the basis of a constant velocity is performed. The profile must be stored as a 3D-file (see also option Open 3D-file under 3D-File Menuitem) and must represent a so-called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source".

The **diffraction stack** is done in the x-t range, this means that an unweighted summation for each point of the profile over a calculated hyperboloid of preset bandwidth is performed. The bandwidth means the area in x-and y-direction for summation in the given distance dimension (parameter **summation area**) over which shall be summed. A larger area naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbola that extend over a large trace range.

Additionally the **velocity** has to be specified in the dimension predefined by the distance and time dimension. An overview over the optimal velocity can be obtained within the option velocity adaptation where the calculated hyperboloid can be optimally adapted to existing diffraction hyperbola by interactive continuous variation of the velocity. The precondition for good determination of the velocity is the existence of diffraction arrivals. Here, also the success of the migration can be realized best. Ideally they should shrink to a very small area. Beside the contraction of the diffraction energy another aim of migration is to shift the arrivals to their "true" position. This is important for steep dipping reflectors. The time range for applying the migration may be restricted. The filter parameters **start time** and **end time** define the time range.

The radio groupbox allows you to specify whether the summation is performed using the complete **3D hyperboloid** or whether the summation is only done along the x- and y-axis (parameter **sum over x-y**).

The migration is one of the most important filters. A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions are present, the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

1.11.9.12 3D-Kirchh. migration

A 3D timemigration of a three-dimensional profile on the basis of a constant velocity is performed. The profile must be stored as a 3D-file (see also option Open 3D-file under 3D-File Menuitem) and must represent a so-called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source".

The **Kirchhoff migration** is done in the x-t range, this means that a weighted summation for each point of the profile over a calculated hyperboloid of preset bandwidth is performed. Before the summation the original profile is differentiated. The bandwidth means the area in x-and y-direction for summation in the given distance dimension (parameter **summation area**) over which shall be summed. A larger area naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbola that extend over a large trace range.

Additionally the **velocity** has to be specified in the dimension predefined by the distance and time dimension. An overview over the optimal velocity can be obtained within the option velocity adaptation where the calculated hyperboloid can be optimally adapted to existing diffraction hyperbola by interactive continuous variation of the velocity. The precondition for good determination of the velocity is the existence of diffraction arrivals. Here, also the success of the migration can be realized best. Ideally they should shrink to a very small area. Beside the contraction of the diffraction energy another aim of migration is to shift the arrivals to their "true" position. This is important for steep dipping reflectors. The time range for applying the migration may be restricted. The filter parameters **start time** and **end time** define the time range.

The radio groupbox allows you to specify whether the summation is performed using the complete **3D hyperboloid** or whether the summation is only done along the x- and y-axis (parameter **sum over x-y**).

The migration is one of the most important filters. A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions are present, the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

1.11.9.13 3D-fk migration

A 3D timemigration of a three-dimensional profile on the basis of a constant velocity is performed. The profile must be stored as a 3D-file (see also option Open 3D-file under 3D-File Menuitem) and must represent a so-called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source".

The method works in the frequency-wavenumber (fk) range (see also fk spectrum). Within the fk-range a variable transform is done based on the entered constant **velocity** (frequency is transformed onto the vertical wavenumber). If only a small data volume in time direction is available one should use an interpolation for this transformation. Such an interpolation is not very stable. That is why such an interpolation is not used in the program. This means the data volume in the f-k-range in direction of the variable to be transformed (in this case the frequency) must be large enough. That is why the Stolt-migration should only be used if the number of wavelengths inside the time window to be considered exceeds a critical value (about 10-15). If this does not hold true, the time window must be enlarged. This is done if the filter **scaling factor** is set to a value larger than 1. This factor allows to automatically exceed the time range by the given value (e.g. enter a factor 2: the time range is doubled and therefore the number of wavelengths within this window).

The radio groupbox allows you to specify whether the semblance is used during the migration process in addition.

With **semblance** activated the semblance for each datapoint within the 3D-zero-offset section based on a summation over a calculated hyperboloid. The semblance analysis is done in the x-y-t range, this means that an unweighted summation for each point of the profile over a calculated hyperboloid of preset bandwidth is performed. The bandwidth means the area in x- and y-direction for summation in the given distance dimension (parameter **semblance width**) over which shall be summed. A larger area naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbolas that extend over a large trace range. In addition you must specify the **nom.frequency** in the given frequency dimension.

First the x-t data are transformed into the f-k-range (see also option fk-filter). After having done the transformation the migration process will be done and after that the back transformation into the x-t-range is performed.

In comparison to the Kirchhoff migration one has not to enter a summation width but the "summation" is always done using all traces. Therefore the Kirchhoff migration often gives better results if a restriction of the summation width is desired (e.g. if the diffractions only extend over a few traces). The fk migration is mostly preferable for the migration of extended or steep reflectors.

The migration is one of the most important filters. A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions are present, the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example.

1.11.9.14 3D-Kirchh. 2D-vel.

A 3D timemigration of a three-dimensional profile on the basis of a 2-dimensional velocity distribution is performed. The profile must be stored as a 3D-file (see also option Open 3D-file under 3D-File Menuitem) and must represent a so-called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source". The 2D-dimensional velocity distribution is given along the hypothetical total distance axis of the 3D-file (summed up distance along the 2D-cuts in profile direction). Therefore it is possible to enter a velocity field for the complete 3D-range but you must consider that the interpolation is done along the hypothetical distance axis.

The **Kirchhoff migration** is done in the x-t range, this means that a weighted summation for each point of the profile over a calculated hyperboloid of preset bandwidth is performed. Before the summation the original profile is differentiated. The bandwidth means the area in x-and y-direction for summation in the given distance dimension (parameter **summation area**) over which shall be summed. A larger area naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbola that extend over a large trace range.

The radio groupbox allows you to specify whether the 2D-velocity distribution is obtained from the **CMP-analysis** (see also CMP 2D-model) or from the **hyperbola adaptation** (see also velocity adaptation). In addition you may specify if only a 1-dimensional velocity distribution is generated either from a CMP or a hyperbola model (options **CMP-1D-model** or **hyp.1D-model**). In this case only the vrms-distribution (see below) which is given for the smallest distance is taken into account. This restriction drastically fastens the migration because the summation hyperboloids must only be calculated once.

The current VRMS- or average velocity for each point of the profile is determined from the 2D-distribution and the summation is performed with this velocity, i.e. for the calculation of the current diffraction hyperboloid only a constant velocity is assumed. However this is a valid approximation if the diffraction branches do not extend over a large laterally heavily fluctuating area. For Ground Penetrating Radar data this is normally always guaranteed, for seismic data the summation width has to be reduced accordingly in the case of strong lateral velocity fluctuations.

After having started the migration, the desired 2D-velocity distribution going to be the basis of the migration is queried. The 2D-velocity distribution is shown in a separate window.

The time range for applying the migration may be restricted. The filter parameters **start time** and **end time** define the time range.

1.11.9.15 3D-semblance

This option allows to calculate the semblance or the unnormalized correlation for each datapoint within the 3D-zero-offset section based on a summation over a calculated hyperboloid.

The **semblance analysis** is done in the x-y-t range, this means that an unweighted summation for each point of the profile over a calculated hyperboloid of preset bandwidth is performed. The bandwidth means the area in x- and y-direction for summation in the given distance dimension (parameter **width**) over which shall be summed. A larger area naturally increases computing time significantly and is only reasonable for a formation of clearly visible diffraction hyperbola that extend over a large trace range.

Additionally the **velocity** has to be specified in the dimension predefined by the distance and time dimension. An overview over the optimal velocity can be obtained within the option velocity adaptation where the calculated hyperboloid can be optimally adapted to existing diffraction hyperbola by interactive continuous variation of the velocity. Precondition for good determination of the velocity is the existence of diffraction arrivals.

In addition you must specify the **nom.frequency** in the given frequency dimension. This parameter is necessary if the envelope (parameter **envelope** activated within the semblance groupbox) of the semblance shall be calculated. If the nominal frequency is stored within the file header (see also chap. 1.4.3) this value is taken as the default value.

Within the semblance groupbox you also may specify if the semblance data shall be normalized to 1 or to the max. value of the original profile.

1.11.10 fk-filter/data fusion

The following fk processing possibilities are included within the fk-filter/fk-spectrum group.

- fk filter
- fk filter-lineparts
- fk spectrum
- dispersion curve
- data fusion
- mix profile

For some filters available within this group the original trace is shown together with the currently filtered trace, the spectrum of the original trace and the spectrum of the currently filtered trace. The arrows near the original trace and original spectrum are used to enhance the resolution of the trace and spectrum windows. The trace or the spectrum is stretched up and down when using these arrows. The spectrum scale is Hz, if the time scale is in milli-seconds and MHz, if the time scale is in nano-seconds, respectively.

When activating the options fk filter or fk filter-lineparts and after having generated the fk-spectrum a table appears. With all these options the inputs may be entered either interactively within the original profile or using the table input. With the option fk filter or fk filter-lineparts activated first the fk-spectrum must be generated and then the filter parameters can be entered using the table or the interactive input. The options save and load allow to store the current edit values to an ASCII file and to load the wanted edit values from file, respectively.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number**.

The filter parameters for the individual processing steps are entered within the **filter parameter group**.

After having specified the filter parameter, you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing.

Activating the option **sequence proc.** additionally open the Sequence Processing menu. With this menu opened you may easily add the current processing step to the sequence processing flow which may be applied to a selectable number of lines after.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing. Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing. . Exceptions where only one secondary line will be needed:

processing step **mix profile**: here an additional character(s) is asked which will be automatically added to the actual filenames for the creation of the secondary lines which shall be mixed. Example:

actual filename: data_gps.00t - additional characters: _insert -> secondary file:
data_gps_insert

1.11.10.1 Fk filter

This filter acts on the chosen data area. The **fk filter** allows to apply a two-dimension filter defined within the frequency-wavenumber domain to a selected data range.

It is possible to restrict the affected data area that is going to be filtered. For this you must enter within the filter parameter group the **first trace**, the **last trace**, the **first sample** and the **last sample**. It is also possible to define this data area within the original profile. Click on the uppermost left corner of the area to be chosen and drag the mouse to the wanted lowermost right corner with the left mouse button pressed. The filtered data set is restricted to that chosen area. By default the complete data area is set.

After having selected the data range you must activate the option **generate fk-spectrum** (see also fk spectrum) which performs the Fourier transformation of the data range into the frequency-wavenumber domain. Since the two-dimensional Fourier-transformation is very computing time intensive, a window informs the user about the current state of calculations.

After termination of the transformation into the frequency-wavenumber domain the normalized amplitude fk spectrum of the selected data range appears on the original profile image. The y-axis of the calculated fk spectrum is the frequency axis in the given frequency dimension and the x-axis of the fk spectrum is the kx axis in 1/meter. If the horizontal axis of the used data set is not the distance axis or the vertical axis is not the time axis, at the axes of the amplitude spectrum appear the corresponding Fourier transformed physical quantities.

The calculation of the fk spectrum now allows to map the different dips of the events onto different parts of the fk spectrum. Events with downdip to ascending distances are mapped onto the right part of the fk spectrum whereas events with updip are mapped onto the left part. This allows you to easily distinguish between these types of events. In addition nearly horizontal events are mapped along small k values within the middle of the fk spectrum.

From the two-dimensional spectrum you have to select the filter area. This area may be defined manually (option **manual input** activated) or by defining a velocity fan (option **velocity range** activated). The options **neg. kx** and **pos.kx** may be used for an easy definition of a filter range for only negative or positive kx-values.

With **manual input** activated the program demands the input of margin points using the left mouse button within the fk spectrum. Set points are connected with straight lines. Analogous to the other edit inputs the chosen values are automatically transferred to the table input and vice versa to the fk spectrum image. The last set point is automatically connected with the first set point so that a closed curve is produced.

With **velocity range** activated you may define two different fans consisting of 2 different velocities respectively to be defined within the filter parameter group (enter in the given velocity dimension). The chosen velocities are automatically transformed into edit pairs and shown within the fk spectrum. Such a fan allows you to easily define a filter for special dips.

With **single velocity** activated the filter range will be determined from one single velocity. Depending on the settings of bandpass or notchfilter all velocities greater than the entered velocity will be kept (bandpass activated) or removed (notchfilter activated).

With **random noise suppression** activated random noise may be suppressed by raising the amplitude spectrum to an exponential power. The higher the **exponential power** parameter the higher the filter effect. A value of 1 shows no effect. A default value is 1.5.

With the option **bandpass** activated the filters acts as a bandpass, this means that the defined area remains unchanged whereas in the outer area the values are set to 0.

With the option **notchfilter** activated the filters acts as a notchfilter, this means that the defined area is set to 0 whereas the outer area remains unchanged.

In addition you may specify a taper. Analogous to the bandpassfrequency filter sharp steps at the filter borders may cause unwanted filter effects. The

FKSpectrumTaper is used to suppress those unwanted effects. You may choose between the **none** (no taper), **linear** (linear decrease from the border with the given taper length), **Hanning** (cos-window) or **Hanning**2** (cos**2-window). The width of the taper both in f and k direction is controlled by the options **taper width (f)** and **taper width (k)** both given in points. The taper points into the filter direction, this means for bandpass outside the chosen bandpass region and for notchfilter inside the chosen notchfilter region.

The option **save** stores the current edit values on an ASCII file under the project directory with the name **fkfil** and a freely selectable extension. Each line of the ASCII file consists of two columns: 1. column: kx value in 1/meter, 2. column: frequency in the given frequency dimension.

The option **load** loads the wanted edit values from an ASCII file with the filename **fkfil**. The table and **fk spectrum** image are automatically updated.

The option **reset** resets the current edit values.

A precondition for the application of the **fk filter** are equidistant traces. In order to save computing time it is recommended to compress the data before using this filter both in x- and in t-direction to that extent that is possible without causing aliasing. However, when applying the **fk filter** the amplitude spectrum of a compressed data set should be checked for spatial or temporal aliasing. After filtering the data set can be re-expanded in order to guarantee a better viewing of the data.

The **fk filter** can be used - though computing time intensive - as lowpass, highpass, bandpass or notch filter. It is highly effective for the suppression of noise with a certain dip in the data set. Like this, horizontal noise signals, as caused for example by reverberations, can be suppressed by cutting the wavenumber zero from the filter area. In an analog way correlated signals with desired gradients can be emphasized.

1.11.10.2 Fk filter-lineparts

The **fk filter-lineparts** allows to apply a two-dimension filter defined within the frequency-wavenumber domain to the complete section which is automatically subdivided into single parts with uniform size and afterwards all individual filtered data parts are composed together into one file.

First you must define the **trace number** within the filter parameter group. This number defines the size of a single data part. For each data part an independent fk filter is performed and afterwards all filtered data parts are put together.

After having chosen the trace number you must generate the fk spectrum and afterwards you have to define the fk filter area analogous to the option fk filter. All other options within the fk filter option are also available.

Using this option is useful if the fk filtering of the complete section exceeds the ram memory of your computer.

Another important application of the fk filter-lineparts is in the case all shots of a seismic experiment for example all combined together within one file. If you are applying the "normal" fk filter on the complete dataset you will get unwanted filter effects resulting from the discontinuous data steps at the shot changes. In this case you should use the option fk filter-lineparts with a trace number equal to the number of traces per shot. A precondition for that is that the number of traces per shot within the data file is always the same.

1.11.10.3 Fk spectrum

This processing step acts on the chosen data area. The processing step allows to calculate the fk spectrum (frequency-wavenumber spectrum) for a definable data area.

It is possible to restrict the affected data area for calculating the fk spectrum. For this you must enter within the filter parameter group the **first trace**, the **last trace**, the **first sample** and the **last sample**. It is also possible to define this data area within the original profile. Click on the uppermost left corner of the area to be chosen and drag the mouse to the wanted lowermost right corner with the left mouse button pressed. The resting data part will not be used for the calculation of the fk spectrum. By default the complete data area is set.

A 2D section is both a function of time and of space (distance). The Fourier dual of the time is the frequency, the Fourier dual for the space is the spatial frequency or wavenumber. Whereas the frequency is a count of the number of peaks within a unit time, the wavenumber of a dipping event is determined by counting the number of peaks within a unit distance along the horizontal axis. The y-axis of the calculated fk spectrum is the frequency axis in the given frequency dimension and the x-axis of the fk spectrum is the kx axis in 1/meter.

The calculation of the fk spectrum now allows to map the different dips of the events onto different parts of the fk spectrum. Events with downdip to ascending distances are mapped onto the right part of the fk spectrum whereas events with updip are mapped onto the left part. This allows you to easily distinguish between these types of events. In addition nearly horizontal events are mapped along small k values within the middle of the fk spectrum.

The fk spectrum is often used to define a spatial filter in order to emphasize events having special dips (see also fk filter).

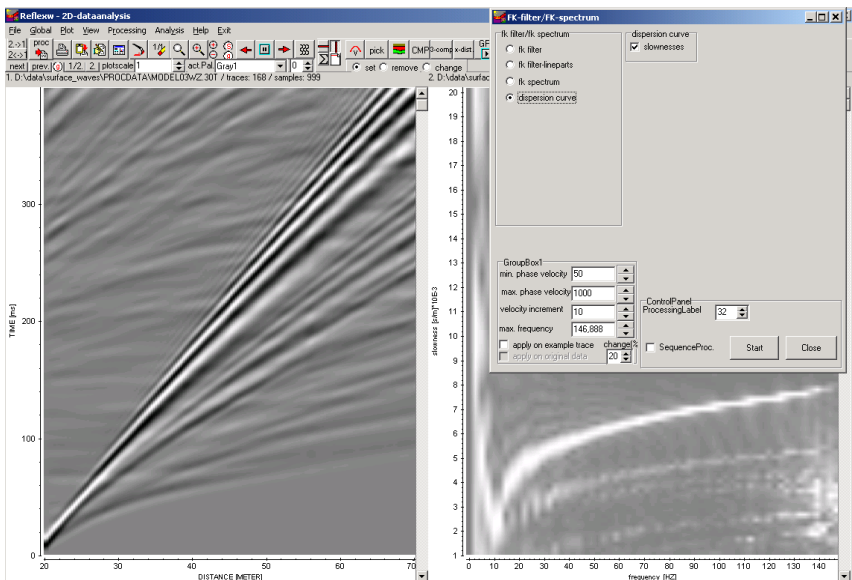
1.11.10.4 dispersion curve

This processing step acts on the chosen data area. The processing step dispersion curve allows to calculate the dispersion curves of a single multi-channel record. The output is either within the frequency-phase velocity (f-v) or frequency-phase slowness (f-p) domain (option slownesses activated).

As input parameters the **min. and max. phase velocities** as well as the **velocity increment** and the **max. frequency** must be entered. Default values are given for the phase velocity window. The default value of the max. frequency will be automatically determined from the data or from the nominal frequency given within the fileheader (3 * nominal frequency).

The option **slownesses** controls if the dispersion curve output is within the f-v or f-p (option activated) domain. In the f-p domain the data normally shows less smearing at lower frequencies. Different algorithms are used for the two domains - therefore no interpolation is necessary for a subsequent transformation.

The dispersion curve output can be used to analyze the phase velocity frequency behaviour of a single shot record. Picking the fundamental or higher modes allows to invert a velocity depth model (normally shear wave) within a subsequent step using an appropriate method.



1.11.10.5 data fusion

This filter acts on the chosen data area. The option allows the fusion of a dual frequency data acquisition. The two datasets of different signal frequencies must have been acquired using the same tracincrement and the same start and end-coordinate. The two datafiles must have the same filename except the channel number.

Normally the channel number is given within the last place of the filename: BAS__007_00.DAT and BAS__007_01.DAT

Exception for IDS-data: Here the channel number is given within the fourth place of the filename:

LA010001.DAT and LA020001.DAT

The first channel must be chosen for the IDS-data.

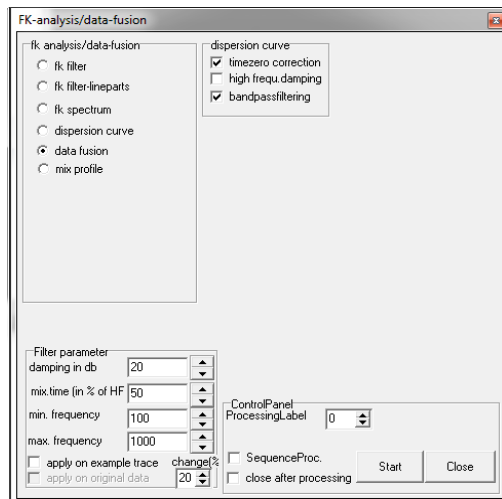
If the measured times are different the file with the larger time should be chosen (not for IDS-data) or the measured time should be enlarged using the processing option time cut before.

The program automatically differs between low and high frequency data either by calculating the mean frequency or by taking the value given within the fileheader. It may happen that the automatic determination of the mean frequency fails and the low and high frequency datasets will be interchanged. In this case the data fusion does not work correctly. Therefore it is recommended to save the mean frequency within the fileheader of each file first. This may be done for example within the edit several file headers menu.

The data must be time zero corrected. This can be done automatically within this processing step if the option **timezero correction** has been activated. The method uses the processing option correct max.phase with the suboptions first onsets, zero cross., use mean, auto correct and a threshold of 20 % of the max. amplitude. A possible DC-shift or low frequency dewow must have been removed before.

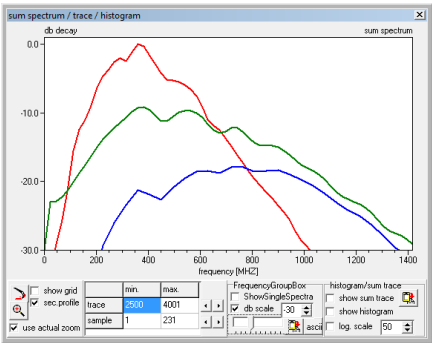
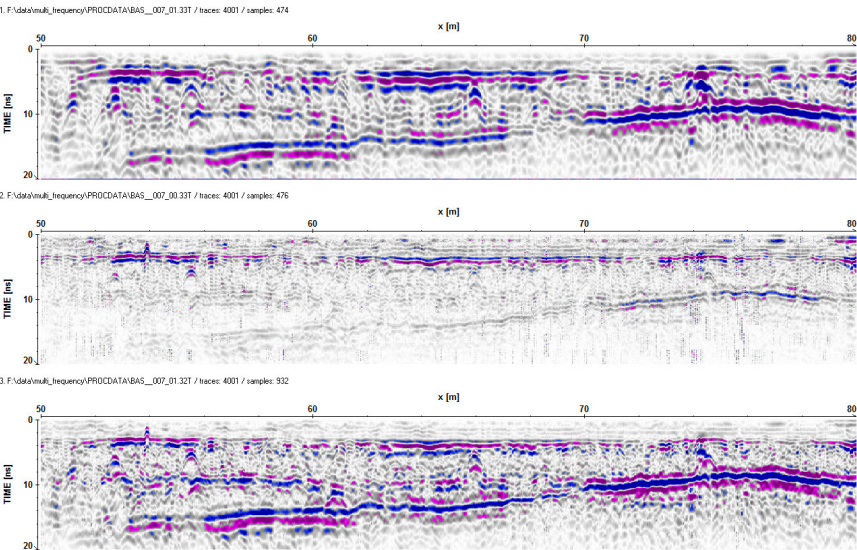
A mixing time must be entered in terms of the percentage of the measured time of the high frequency (HF) data (**option mix.time/in % of HF**). The low frequency data will be damped until this mixing time starting with a value given by **damping in db** at time 0 using a linear interpolation. If the option **high freq.damping** is activated the high frequency data will also be damped starting at the mixing time and ending at the HF measured time again using a linear interpolation and using the damping in db at the HF measured time.

Optionally a **bandpassfiltering** will be done using the **min./max. frequency** parameters. Such a filtering is recommended because the mixing normally introduces some high frequency noise which should be removed using a low pass filtering.

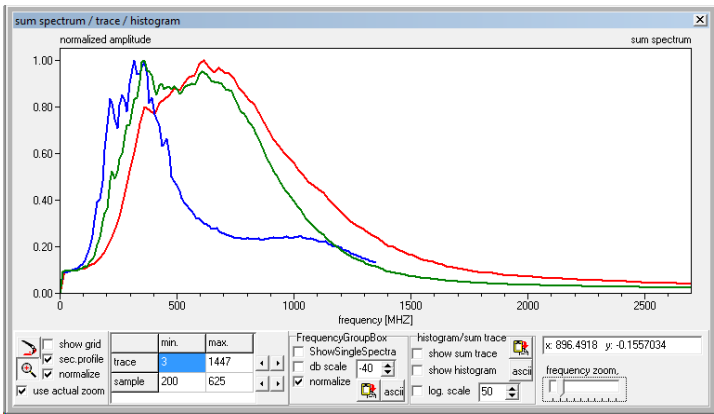
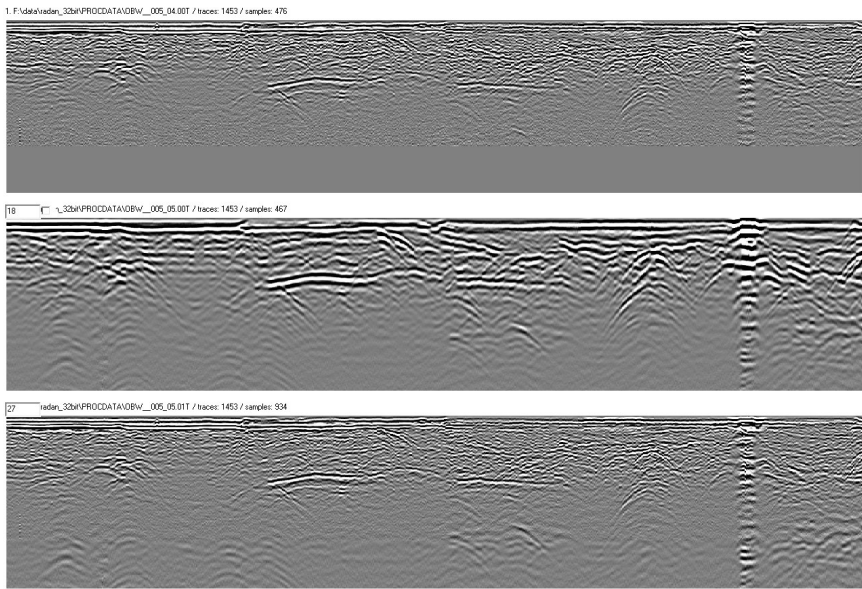


The option may be used within the sequence processing but the option data fusion must stand at the first place within the processing flow. This is necessary because the option fuses two different datasets into one and therefore only subsequent dataprocessing steps (e.g. background removal, migration) are allowed.

The following picture shows an example of a two frequency acquisition. The upper picture shows the lower frequency data, the middle the high frequency and the lower picture the fused data always together with the spectra. The frequency spectra are displayed within a separate window (red 300 MHz antenna, blue 800 MHz antenna, green fused dataset).



Following you find another example with a stronger focus on the high frequency data which are displayed within the upper window (red frequency curve). The middle window shows the low frequency data (blue frequency curve) and the lower one the fused data (green frequency curve).



1.11.10.6 Mix profile

This option gives the possibility to **add**, **subtract**, **multiply**, **divide** or **overlay** a second profile acting on the primary file. The precondition for this is that the two profiles have an identical number of traces and of samples. The distance or depth values written in the trace header are averaged, if the values of both sections do not agree.

The mixing for add and subtract will be done either within **time** or **frequency** range to be defined within the **working range** panel.

The option **load** within the SecondLineGroupBox enables the opening of the file which is to be added, subtracted, multiplied, divided or overlaid.

The options within the MixProfileGroupBox allow you to define which function shall be used.

The option is not available within the Sequence Processing.

The option is available for the **batch start** processing. Here an additional character(s) is asked which will be automatically added to the actual filenames for the creation of the secondary lines which shall be mixed. Example:
actual filename: data_gps.00t - additional characters: _insert -> secondary file:
data_gps_insert

1.11.11 Deconvolution/shaping

The following deconvolution possibilities are included within deconvolution/shaping group.

- deconvolution
- deconvolution/shap.
- Spectral whitening
- differentiation
- integration
- shaping
- extract wavelet

For some filters available within this group the original trace is shown together with the currently filtered trace, the spectrum of the original trace and the spectrum of the currently filtered trace. The arrows near the original trace and original spectrum are used to enhance the resolution of the trace and spectrum windows. The trace or the spectrum is stretched up and down when using these arrows. The spectrum scale is Hz, if the time scale is in milli-seconds and MHz, if the time scale is in nano-seconds, respectively.

When activating the options fk filter or fk filter-lineparts and after having generated the fk-spectrum a table appears. With all these options the inputs may be entered either interactively within the original profile or using the table input. With the option fk filter or fk filter-lineparts activated first the fk-spectrum must be generated and then the filter parameters can be entered using the table or the interactive input. The options save and load allow to store the current edit values to an ASCII file and to load the wanted edit values from file, respectively.

The trace number for the trace to be displayed in the trace and spectrum windows can be chosen using the option **trace number**.

The filter parameters for the individual processing steps are entered within the **filter parameter group**.

After having specified the filter parameter, you must enter the processing label (option **ProcessingLabel**). If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the processing.

Activating the option **sequence proc.** additionally open the Sequence Processing menu. With this menu opened you may easily add the current processing step to the sequence processing flow which may be applied to a selectable number of lines after.

The option **start** starts the processing using the actual settings.

The option **batch start** starts a batch processing of the actual processing step for a selectable number of profiles. The actual settings are used for the processing. Some processing steps which need the loading of secondary files are not supported equivalent to the sequence processing.

1.11.11.1 Deconvolution

The filter acts on each trace independently. The filter permits the application of a deconvolution on the profile. This is a so-called spiking-deconvolution, i.e. it is tried to create a highly broadband and smooth spectrum (according to the spectrum of a spike). The program uses the recursion-algorithm of Levinson (Wiener-Filter method). For each trace the autocorrelation over a predefinable number of samples is calculated and used as input signal for the Levinson-Algorithm, which calculates recursively the deconvolution-filter trace. Then the filter trace is convolved with the original trace.

The deconvolution altogether needs four parameters (beginning and end of the autocorrelation range, length of the filter trace, portion of white noise in percent). With the specification of the beginning (**autocorr.start**) and the end of the range for the autocorrelation (**autocorr.end**) a certain control of the effects of the deconvolution is possible. For a suppression of multiples for example it is necessary to choose a large time range for the autocorrelation, whereas a small range is sufficient when "only" the original signal should be transformed back into a spike.

The length of the filter trace (parameter **filter length**) normally can be kept small. It must be smaller than or equal to the autocorrelation range.

To stabilize the deconvolution process white noise (parameter **noise (perc.)**) has to be added to the data. The input parameter noise (percent) determines the portion of white noise. A value that is too high leads to an insufficient performance of the deconvolution, a too low value leads to stability problems.

1.11.11.2 Deconvolution/shap.

This statistical deconvolution is based on the recursion-algorithm of Levinson (Wiener-Filter method) for systems of linear equations with Toeplitz structure. The filter acts on each trace independently. For each trace the autocorrelation over a predefinable number of samples and the crosscorrelation between the input and the desired output are calculated.

These correlations are the input of the Levinson-Algorithm, which calculates recursively the deconvolution-filter trace. Then the filter trace is convolved with the original trace.

Depending on the goal of the deconvolution the desired output is selected. The following types of statistical deconvolution are implemented:

spiking-deconvolution:

The goal of spiking-deconvolution is a concentration of the signal-wavelet in the time domain or a broad and smooth spectrum of the deconvolved trace in the frequency domain. The desired output is a spike, i.e. a wavelet with maximum energy concentration. If **0 lag spike** is selected the energy is concentrated at the beginning of the wavelet, if **spiking** is selected the energy is concentrated at the sample defined by the parameter **lag**. In practice the input signal is mixed-phase and the best spiking deconvolution is done with a lag corresponding to the time of the maximum energy in the input signal wavelet. As spiking deconvolution flattens the amplitude spectrum the deconvolution may cause high frequency

noise and a band-pass filtering after deconvolution is necessary. This can be done implicitly by choosing the parameter **filter spectrum**. With this option the amplitude spectrum of the filtered trace is scaled by the amplitude spectrum of the unfiltered trace.

wavelet shaping:

The goal of wavelet-shaping is to form the wavelet of the output signal e.g. to improve the minimum-phase properties of the output signal wavelet. The deconvolution type **any signal** allows to load the new wanted waveform from an independent Reflexw datafile (see also processing option shaping, chap. 1.11.1.15). The deconvolution types **zero phase** and **min. phase** transform the input signal wavelet to the corresponding zero-phase or minimum phase wavelet. In practice the difference in transforming to a zero-phase wavelet or a minimum phase wavelet is often small. As transforming to a minimum phase wavelet needs more computation time we prefer the deconvolution type zero phase. In practice the input signal is mixed-phase and the best deconvolution is done with a lag corresponding to the time of the maximum energy in the input signal wavelet.

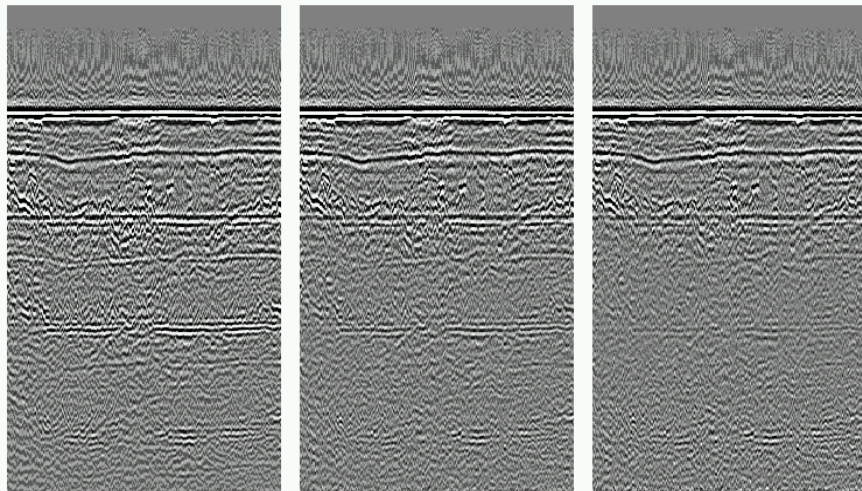
predictive deconvolution:

The goal of predictive deconvolution is the suppression of multiples or reverberations. The desired output is a time advanced (parameter lag) version of the input signal. To suppress multiples choose a lag corresponding to the two-way-traveltime of the multiple, to suppress reverberations choose a small lag corresponding to the duration of the essential reverberation. If the input signal is mixed-phase a spiking deconvolution or wavelet shaping may improve the result of the following predictive deconvolution.

If the signal shape varies strongly from trace to trace the deconvolution may reduce the spatial coherences of reflections. This is caused by strong changes of the autocorrelation from trace to trace. To overcome this problem use the parameter **mean auto**. With this parameter set the mean of all trace-autocorrelations is used to calculate the deconvolution-filter. All traces are deconvolved with the same deconvolution-filter and spatial coherences will maintain.

The deconvolution altogether needs three parameters (beginning and end of the autocorrelation range, length of the filter trace). With the specification of the beginning (**autocorr.start**) and the end of the range for the autocorrelation (**autocorr.end**) a certain control of the effects of the deconvolution is possible. For a suppression of multiples for example it is necessary to choose a large time range for the autocorrelation, whereas a small range is sufficient when the original signal should be transformed into a spike. The length of the filter trace (parameter **filter length**) normally can be kept small. It must be smaller than or equal to the autocorrelation range. The lag (parameter lag) must be smaller than the filter length. The parameters autocorr.start, autocorr.end, filter length and lag are given in the current time dimension.

The option **runs** allows to apply the deconvolution several times using the same parameters. The picture on the left shows the original data showing distinct multiples from the surface. The middle picture shows the result after one predictive deconvolution, the right panel shows the result after 7 runs.



1.11.11.3 Spectral whitening

The filter acts on each trace independently. The filter **spectral whitening** allows to compensate for frequency attenuation with time. The aim of the filter is to flatten the spectrum for a given frequency band (filter parameter **start frequency** and **end frequency**).

The radiogroup **spectral whitening** controls the method for the spectral compensation, with the options **add derivatives** and **norm frequ.bands**.

With **add derivatives** activated the spectral enhancement is done by adding the second, fourth and sixth-order derivatives as well as a smoothed trace.

With **norm frequ.bands** a series of narrow overlapping band-pass filters is applied on each trace. The same series of amplitude decay curves is then computed to describe the decay rates for each frequency band. The inverses of these decay curves are applied to each bandpass filtered trace and the results are summed. In order to preserve the amplitude characteristics of the original trace the summed trace is then multiplied with the amplitude decay curve of the original trace.

The filter changes the overall amplitudes. Therefore a **scaling** factor allows to rescale the filtered traces with a given value.

The spectral whitening allows to compensate for frequency attenuation with time. It also partially deconvolves the effective source wavelet (see also deconvolution).

1.11.11.4 differentiation

The filter acts on each trace independently. The filter **differenatiation** allows to calculate the derivative in time for each trace.

The parameter **degree of differ.** defines the order - 1 means 1. derivative, 2 2. derivative, and The parameter **norm. (0=yes, 1=no)** defines whether a normalization of the amplitudes shall be done. If the parameter is set to 0 the amplitudes after the differentiation will be normalized in order to ensure that the resulting amplitudes are within the same amplitude range as the original data. If set to 1 no normalization will be done and the amplitudes represent true values.

The calculation of the derivative is performed in the frequency domain by multiplication by the frequency.

The filter 1. derivative allows to emphasize the high frequency part of the individual signals and therefore may enhance the vertical resolution under special conditions (see also deconvolution).

1.11.11.5 Integration

The filter acts on each trace independently. The filter **integration** allows to integrate the signals in time for each trace.

The parameter **degree of integrat.** defines the order - 1 means single integration, 2 means double integration and The parameter **norm. (0=yes, 1=no)** defines whether a normalization of the amplitudes shall be done. If the parameter is set to 0 the amplitudes after the integration will be normalized in order to ensure that the resulting amplitudes are within the same amplitude range as the original data. If set to 1 no normalization will be done and the amplitudes represent true values.

The integration is performed in the frequency domain by division by the frequency.

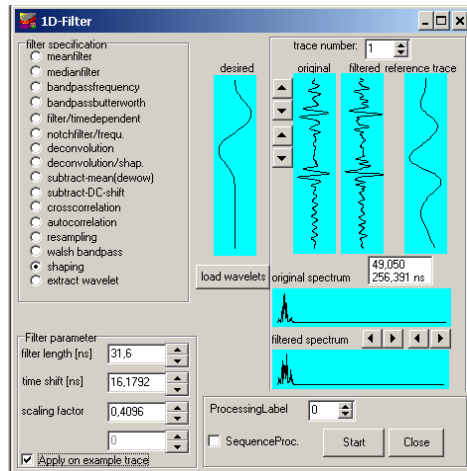
The integration allows to emphasize the low frequency part of the individual signals (see also complex trace-analysis (envelope)).

1.11.11.6 Shaping

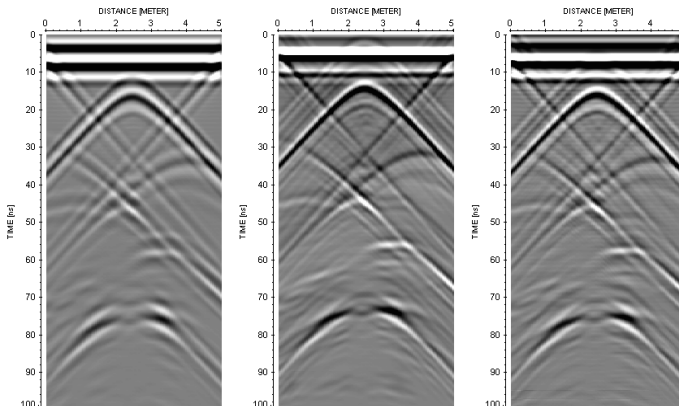
The option allows to convert the characteristic waveform of the profile to a new desired one.

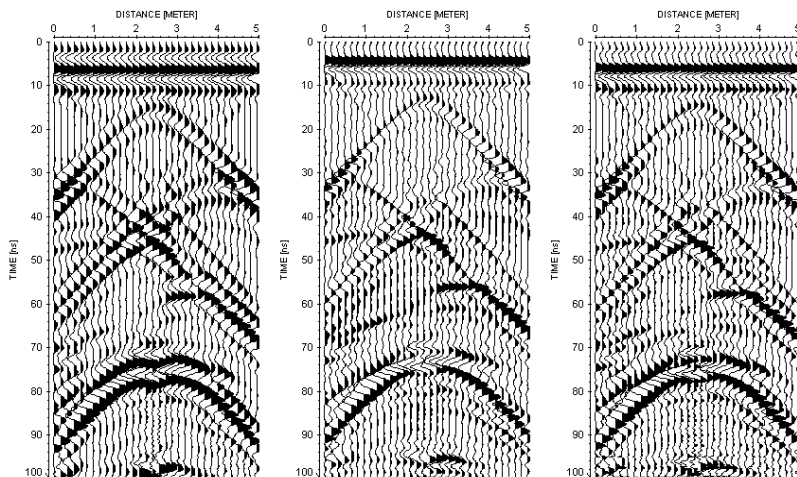
Both the new desired waveform and the characteristic input wavelet must exist within an independent Reflexw datafile. The two Reflexw datafiles may contain one or more traces but in any case only the first trace will be taken as the corresponding new desired and old input wavelet. Such a wavelet can be easily extracted from any profile using the option **extract wavelet** (chap. 1.11.1.16). After having chosen these desired and input waveform file the **filter** **length** and the **time shift** must be entered. By default the filterlength and the timeshift equal the length of the desired wavelet. The **scaling factor** allows to enter a multiplication factor by which the data are multiplied after shaping. The amplitudes of the desired and the characteristic waveform should be in the same order.

The option **load wavelets** allows to reload the desired and input wavelets.



The correct choice of the filter length may reduce the effect of filter induced precursors. The following two pictures (point and wiggle mode) show an example. On the left panel the original data are shown. The middle panel shows the result using a larger filter length than the length of the desired wavelet and





the right panel shows the result with a filter length equal to the desired wavelet length.

The shaping filter can be used for example to reduce the length of the waveforms (e.g. due to reverberations).

1.11.11.7 extract wavelet

The option allows to extract a characteristic wavelet from the actual profile.

The desired characteristic wavelet must be defined by the selection of a trace/time given by the filter parameters. The first filter parameter specifies the start time (**start ref.time**) and the second one the end time (**end ref.time**). The third parameter gives the start distance range (**start ref.distance**) and the fourth one the end distance range (**end ref.distance**). Over the selected traces an automatic averaging is performed for each selected time value. The selected trace/time range can be for example the area of the first arrival.

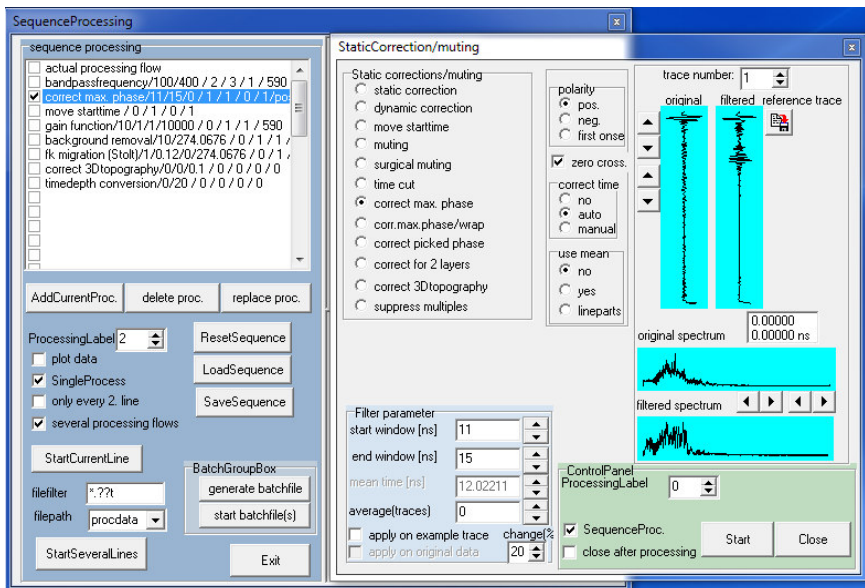
The option can be used for example for defining the new wanted and the input wavelet for the shaping filter (chap. 1.11.1.15).

1.11.12 Sequence Processing

The menu **Sequence Processing** allows a sequential processing, i.e. a selectable number of lines is processed with an identical sequence of processing steps. For this you have to define a sequence of processing steps before, then select the data lines and then start the sequence processing.

The **current processing flow** is shown in the uppermost window. You can mark the individual processing steps. Mark a processing step if you want to delete it or if you want a new processing step to be inserted before. You also have the possibility to change the sorting of the individual processing steps by simply dragging one processing step to another place (pressed left mouse button).

A direct **control** and **modification** of a sequence processing flow has been included. A double click on the wanted processing step within the sequence processing table automatically opens the corresponding processing menu showing all entered parameters of the processing step. This can be used either for a control or for a modification. Change the parameters and click on replace proc. - this allows you to change the settings of the actual processing step within the sequence processing flow.



The definition of the processing flow is done with the options **AddCurrentProc.**, **DeleteProc.**, **ResetSequence**, **LoadSequence** and **SaveSequence**.

The option **AddCurrentProc.** adds the current filter together with the filter parameter to the processing flow. A precondition to that is that the sequence processing menu is opened together with any filter menu. The current filter is added at the end of the processing flow if no processing step within the processing flow is marked. If any processing step is marked, the current filter is inserted before this processing step.

The option **delete proc.** removes the marked filter step.

The option **replace proc.** replaces the marked filter step by the current filter. The option works according to the option **AddCurrentProc.** but the marked filter step will be replaced.

The option **ResetSequence** resets the current processing flow to zero.

The option **LoadSequence** loads a processing flow previously save from disk.

The option **SaveSequence** saves the current processing flow on disk. The file automatically receives the extension PFL and is saved in the project directory.

The **ProcessingLabel** defines the extension of the output datafile names. If already a data set with the same name exists, i.e. also with the same label, it is overwritten after starting the sequence processing.

With the option **plot data** activated the current original file together with the current processed file are plotted during the sequence processing. Deactivate this option if plotting takes too much computer time.

The options **Single Process** and **Intermediate Process** control how the sequence processing is done:

With **Single Process** and **Intermediate Process** deactivated the program executes the processing steps sequentially on the selected profiles, i.e. the 1st processing step is applied to the original profile. To the profile processed in that way then the next step is applied and so on. As a result only one processed file will be created from each original file. The intermediate processing steps will not be saved.

With **Single Process** activated each processing step is independently applied on the original line, i.e. the 1st processing step is applied to the original profile, then the 2nd processing step is applied to the original profile and so on. This results in different output lines. The entered processing label is used for the first processing step for all chosen lines. The label for the other processing steps is incremented by 1 each time. You have chosen for example three processing steps and specified the number 10 as processing label. The processing labels 10, 11 and 12 for the three different steps are used. Notice that possibly existing files are overwritten.

With **Intermediate Process** activated each intermediate processing step is stored separately. This results in different output lines. The entered processing

label is used for the first processing step for all chosen lines. The label for the other intermediate processing steps is incremented by 1 each time. You have chosen for example three processing steps and specified the number 10 as processing label. The processing labels 10, 11 and 12 for the three different intermediate steps are used. Notice that possibly existing files are overwritten.

With the option **only every 2. Line** the processing will be restricted to every second chosen line starting with the first chosen line. This option might be useful for example if each second line shall be flipped. This option simplifies the file choice process which is quite expensive for every second line if the filenames are in ascending order.

With the option **several processing flows** activated several processing flows may be applied in addition to the actual one consecutively. Depending on the number of additional processing flows a corresponding number of processing files will be produced. The entered processing label is used for the actual processing step for all chosen lines. The label for the additional processing flows is incremented by 1 each time. The option might be useful if a special processing step must stay at the first position within the processing flow, e.g. the normalize profiles option. The option is only available for the sequence processing of several lines (option StartSeveralLines).

The option **StartCurrentLine** allows to apply the current processing flow on the current original file loaded within the 2D-dataanalysis.

The option **StartSeveralLines** allows to apply the current processing flow to several lines. **Filefilter** allows to enter a filter for the multi file choice. **Filepath** specifies the path for loading the files to be processed. The maximum number of files is restricted by the system (about 1000 files). This number is not exact and therefore no warning message appears. If the **filetype** within the openfile-menu is set to **ASCII-table** the filenames of the wanted datafiles are read from a separate ASCII-file. Each line of the ASCII-file contains the name of the datafile without the filepath. The complete filename of each datafile is given by the current project directory, the chosen filepath (procdata or rawdata) and the filename stored within the ASCII-file. This option overcomes the problem with specifying more than about 1000 files.

Within the **BatchGroupBox** you may define and perform a batch processing, this means several sequence processings within one step.

The option **generate batchfile** creates an ASCII file with the extension SEQ under the current project directory containing:

1. line: *directory name for the datafiles to be processed*
2. line: *sequence processing datafilename*
3. line: *processing label*
4. -n. line: *filenames*

example:

```
d:\reflex\demo\procdata
d:\reflex\demo\mark1.pfl
0
file001.01t
file002.01t
```

file003.01t

After having activated the option you are asked for the wanted sequence processing filename and the sequence processing label. Then you have to select the wanted datafiles.

The option **start batchfile** starts the batchprocessing. After having activated the option you must select the wanted ASCII batchfiles. The chosen files are alphabetically resorted (sorted with ascending order). Then the batch processing is started.

To be considered: the different ASCII batchfiles must be located under one single directory but the ASCII batchfiles may contain any working directory (see line 1 within the ASCII batchfile).

Peculiarities:

The following processing steps are not supported by the sequence processing: extract, replace, duplicate, move, insert profile, add profile, subtract profile and combine files f.CMP.

The sequence processing flow from the DOS-version of REFLEX cannot be used anymore because the names of the processing steps have been changed.

When using the processing step interpolate only the interpolation of periodic groups based on the set filter parameters is supported.

When using the processing step markerinterpol, the marker positions are always read from the trace headers of the individual original file. You may choose for the markerincrements between:

1. option equidistant - the entered single marker increment is used for all marks.
2. option read from one markerfile - the markerincrements are read from one single ASCII-file which has been created using the option save under markerinterpol. The markerincrements are the same for all profiles.
3. option read from profile markerfile - the markerincrements are read from a separate ASCII-file for each profile. The filename of the ASCII-file corresponds to the profile filename (without extension) and must have been stored directly under the projectdirectory and must have the extension mar. Example for 2 profiles:

c:\data\procddata\file01.00t C:\data\file01.mar

c:\data\procddata\file02.00t C:\data\file02.mar

For the following processing steps the edit values are read from a file: manual gain (y), manual gain (x), static correction, muting and fk-filter. If any of these processing steps is included within the current processing flow the wanted file must be chosen. The file must have been created in a preceding step within the processing menu using the option store.

plotoptions: the current settings of the plotoptions (point or wigglemode, amplitude scale and so on) are stored within the fileheader of each processed file. If no primary file has been loaded and therefore no plotoptions are defined those plotoptions stored within the fileheader of the first profile to be processed are used.





1.12 Analysis MenuItem


Within this menuitem you may choose between the following analysis possibilities:


Velocity adaptation MenuItem:	interactive velocity adaptation based on diffractions, reflections or lines
Pick MenuItem:	picking of onsets
LayerShow MenuItem:	combine picked onsets and create a report for the individual layers
CMP-processing MenuItem:	With the CMP-processing option activated a complete CMP-processing of single shots stored in one file is possible.
AD-conversion MenuItem:	enter the AD-conversion module for digitize analogue data
edit comment marker MenuItem:	enables the edit possibilities for the comment markers
adjust trace gain MenuItem:	enables an individual gain adaptation for the single traces
sum amplitudes for each trace:	sums all amplitudes (absolute values) for each trace and stores the sum within an ASCII file together with the corresponding distance of the trace. The data can be viewed using the option add. 2 colum data under view.
create MPEG-file:	option allows to create a MPEG moviefile for the later use with a MPEG-player.
2D-lines statistics	allows to to analyse the maximum and the mean of individual 2D-lines as well as the dominant frequency.


1.12.1 Velocity adaptation


The option interactive velocity adaptation allows the interactive adaptation of **diffraction** or **reflexion hyperbola** by calculated hyperbola of defined velocity and width. There is also the possibility of fitting **linear features** either by interactively changing a line or by setting two points. The max. number of different adaptations is 100. The speed button **core** allows to interactively vary the velocities of the single layers of the individual cores. The velocities may be stored on file and may be reloaded at any time. The velocities are combined into a 2D-model by using a special interpolation (see also option x-weight). Such a 2D-velocity distribution may be used in a subsequent step for the migration or the depth conversion.


You may choose between the adaptation of linear features (speed button ) and speed button , of diffraction hyperbola (speed button **D**, the profile must be a zero offset profile) and of reflexion hyperbola (speed button **R**) and of core data (speed button **core**). In addition there is the possibility of an intercepttime inversion for refraction data (speed button ) as well as a direct wave adaptation of VSP-data (up or downhole data, speed button )

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

The speed button  allows the interactive use of the intercept time method for refraction data. The option enables to get a first 1D-model very quickly. After having activated the option you must enter the bending points by the left mouse button. The first point is automatically set to time zero and distance of the shot position. Move the mouse with pressed left mouse button to the next bending point and leave the mouse button and so on. The start point of a new branch always corresponds to the endpoint of the previous one. The 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.

Using the option save  (file with extension HYP) an ASCII-1D-model-file with the extension VEL is automatically saved in addition, which allows an easy access to the interpreted 1D-model. This 1D-model can also be used within the CMP velocity analysis window (chap. 2).

The speed button  allows the interactive adaptation of the direct wave of a VSP (vertical seismics profiling) measurement (uphole or downhole measurements). After having activated the option you must enter the bending points by the left mouse button. The first point is automatically set to time zero and distance of the shot position (normally 0). Move the mouse with pressed left mouse button to the next bending point and leave the mouse button and so on. The start point of a new branch always corresponds to the endpoint of the

previous one. After having finished the settings the depth, velocities and thicknesses of the calculated 1D-model are shown in a window. Using the option  (file with extension HYP) an ASCII-1D-model-file with the extension VEL is automatically saved in addition, which allows an easy access to the interpreted 1D-model. This 1D-model can also be used within the CMP velocity analysis window (chap. 2).

The speed button **core** is only enabled when a core datafile is loaded (see suboption Core data/1D models under View Menuitem). If the core datafile does not contain the velocities of the individual core layers all velocities are preset to the current adaptation velocity. With the speed button **core** activated you may interactively adapt the core bars to layer boundaries visible within the 2D data. Click on any layer bar which you want to adapt. The fill style of this layer bar changes. With the adaptation parameter **v** highlighted it is now possible to change the velocity of this distinct layer by pressing the key < and > respectively or using the mouse wheel (see below). With the option **change all** activated all layer velocities of the actual cores are changed simultaneously. With the adaptation parameter **thickness** highlighted it is possible to change the core thicknesses. Changing the thicknesses might be useful for example for updating a core data file for topographically corrected data. With the option **save vel, 2dm-files** within the global settings menu activated both a vel-datafile for each core and a 2dm-data file for all cores will be automatically generated when saving the cores. The filename is taken from the original datafile name including an extension for the vel-data. Example for 10 cores and original datafile data_wheel.01t: -> data_wheel_001.vel ... data_wheel_010.vel and data_wheel.2dm. These files are identical to the files when created within the CMP-velocity analysis menu and may be used there or for a subsequent migration, time depth conversin,

With any of the other speed buttons activated a hyperbola or a line according to the adaptation type based on the current parameters is calculated and plotted, as soon as the cursor is within the working window of the primary or secondary profile.

Important:

The type **reflexion** adaptation needs CMP- or Moveout-data.

The type **diffraction** needs zero-offset data or a negligible source-receiver distance with respect to the depth of the object. With the option **use S/R-dist.** activated the source-receiver distance stored within the fileheader is taken into account for the calculation of the diffraction hyperbola. This mainly affects the apex of the hyperbola.

By pressing the left mouse button the current diffraction hyperbola is maintained and plotted on the profile. Additionally the current velocity is displayed in a window.

All the adaptation parameters **v**, **width**, **radius** and **angle** can be interactively changed using the keys < and > respectively or using the mouse wheel/trackbar (if the yellow light button is active the trackbar (mouse scroll) for an interactive change of the velocity parameters will always get the focus). The track bar on the right of the adaptation parameters may be used as well. The parameter which can be changed is shown in aqua color. Activate the parameter to be changed

interactively by simply enter it. The size of the interactive change is controlled by the parameter **change(%)** given in percentage.

The input parameters in detail:

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

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

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

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

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

The speed button on the left-hand side allows to **reset** all adaptations, to **load** existing velocity adaptations and to **save** the current adaptations on file with the extension HYP under the rohdata directory.

Using the option **load** it is also possible to load adaptation from an ASCII-file.

Each line of the ASCII-file must have the following form:

profilename tracenummer time velocity

Using the option **save** it is also possible to save the adaptations using an **ASCII-format** with the extension ASC. Each line contains the distance, velocity and time of one adaptation. In case of the intercepttime adaptation each line contains the calculated depth, the velocity, the intercepttime, the distance and the time. In case of the VSP adaptation each row contains the calculated depth, the velocity, the thickness and the time.

The loaded adaptations are plotted at their original profile distance. This means that you have to be careful if you are using adaptations from other, e.g. crossing profiles. When saving the current velocity adaptation, the following parameters are stored: velocity, distance in profile direction, x-weight.

With the option **view on 2. line** activated the marked adaptation are also shown on the secondary line.

Peculiarity for speed button **core**: With the option save corefile under global settings activated the currently loaded corefile is updated using the current layer velocities. With the option save VLA-file under global settings activated a VLA-ASCII velocity datafile (see Create LayerShow velocities) is automatically created which may be used for the generation of a layershow. With the option save vel.2dm-files activated VEL-ASCII velocity datafiles for each core as well as a 2Dm-file (see CMP-velocity analysis) is automatically created when storing the

core velocity adaptations which may be used for e.g. migration, timedepth-conversion, stacking,

Using the option **2D** it is possible to show and save the 2-dimensional velocity distribution based on the current adapted velocities. The interpolation is controlled by the parameter x-weight. If you are using such a velocity distribution in a subsequent processing step (e.g. migration based on 2D-velocity adaptation) again the interpolation is done according to this x-weight parameter. The rasterfile will be stored under the given 2D-filename (under the path rohdata under the actual project directory). If no name has been specified (blank text) a dummy filename veloc0001.dat will be used.

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

1.12.2 Pick MenuItem

With the pick option activated arrivals can be picked. You have the choice between manual picking, continuous pick and a semi-automatic picking using a phase follower.



After having activated the pick option you still have the possibility to activate the velocity adaptation option (chap. 1.12.1) in addition. Thereby for example it is possible to continuously display a calculated diffraction hyperbola during picking for a better determination of the hyperbola cusp.

Picking is always done for the primary profile but you also may pick within the secondary profile and then the pick is displayed within the primary profile. This might be useful if you have acquired a profile with different frequencies and the phase is sometimes better visible on the secondary line. To be considered for continuous picking: the traceincrement as well as the profilestart coordinate must be the same for the primary and the secondary profile if you are picking within the secondary profile. Otherwise a warning message appears.

Corrections can be performed either on zero crossings or on the extremum (option CORRECTION). The picks can be saved with an arbitrary name and reloaded any time (option save and load). The maximum number of picks is limited to the maximum number of scans per profile. If you need more picks for a profile, they must be saved on different files. The picks belonging to one profile can be put together and viewed as a whole with the option layershow and also issued as report on printer or file (see also LayerShow Menuitem).

Both the travel time and the current amplitude of the phase are detected. The manual picking of phases is superior to the automatic picking in respect to time consumption for small data sets and in respect to accuracy for data sets of bad quality. For large data sets it is recommended to have a desired phase semi-automatically picked, first - this is discussed under phase follower, and then to check the quality of the automatically set picks and correct them manually if necessary.

In addition to the distance the xyz-shot- and receiver coordinates stored in the traceheader (see also trace header Edit menu, chap. 1.9.6) of each individual trace are stored within the pick record. These values are used when using the tomography or when exporting the picks to an ASCII-file e.g. in order to display a 2- or 3-dimensional contour plot of the pick values. If you want to create such an ASCII-file you must ensure that the xyz-shot and receiver coordinates are correct. These traceheader-coordinates may not always correspond to the fileheader-coordinates and therefore you have different possibilities to update them (see trace header Edit menu, chap. 1.9.6 and Edit several FileHeaders menu, chap. 1.10).

In addition a velocity is stored with each pick. This velocity may be used for a subsequent depth conversion when saving the picks as ASCII-data.

Within the global settings menu (chap. 1.2.1) you can choose between two different symbols for the picks (* and -). Here also some other pick paramters can

be entered. The size of the pick-symbol can be changed within the symbol font menu (see also Global MenuItem).

manual pick: With the option **set** activated a pick can be set with the left mouse button. The same applies to the manual removal or change of an existing pick after selecting the options **remove** or **change**, respectively. Here those picks are removed or changed which are next to the current cursor position. The right mouse button activates the interpolate option whereby a fast piecewise linear picking is possible. By default the distance position of a pick will not be changed when using the change option. With the option **change pick-pos.** (to be found under add.options) activated it is also possible to change the distance position of existing picks.

continuous pick: With this option activated you can set (option set activated), remove (option remove activated) or change (option change activated) picks by moving the mouse cursor over the data set with continuously pressed left mouse button. The picks are set or removed or changed at those traces that are next to the current cursor position. The continuous picking must be done from left to right, this means with increasing distance coordinates. The option continuous pick therefore makes manual picking a lot easier for large data sets. If the cursor movement is too fast to pick every trace an interpolation between the founded pick position is done. The parameter delay under global settings controls the speed of the continuous picking. The data must be sorted in ascending order with respect to the distances (see also option sort).

To be considered: The continuous picking is done based on the tracenumbers and these tracenumbers are stored with the picks. Therefore problems may occur when you are loading existing picks which have been picked from another dataset with different tracenumbers (this means with different traceincrement and/or startcoordinate). The same holds true for picking within the secondary profile. Again the traceincrement as well as the profilestart coordinate must be the same for the primary and the secondary profile. The right mouse button allows you to toggle between the options set and change. With the plotoption TraceHeaderDistances activated no interpolation is done.

phase follower: The option allows the automatic assignment of picks to a selected phase. For this you first have to select the desired phase at an arbitrary position of the profile with the left mouse button.

With selected secondary option **set** the program then follows automatically this phase in the direction of larger distances. The choice is limited to the displayed screen range. If the phase follower reaches the right end of the screen the profile is scrolled automatically towards larger distances and the pick assignment is not stopped (the option set is also automatically set to change). A precondition for a correct application of this option is that the phase extends continuously over the whole profile without being perturbed by interference effects. The automatically detected picks certainly can be reedited like all other picks.

If the option **change** is activated already existing picks can be changed with the automatic assignment. The program expects like with selected option set the selection of a value within the profile by pressing the left mouse button. If for the selected trace already a pick exists it is replaced by the new automatically assigned pick. With this option it is possible to react fast on an unwanted deviation of the automatic assignment from the desired phase.

The Escape-key as well as the right mouse button allows to abort the automatic assignment.

The speed of the phase follower is controlled by the option **delay** (available within the PickGroupBox if the option **add.options** is activated - see below). Increase this number in order to slow down the phase follower.

With the option **autostop** (available within the PickGroupBox if the option **add.options** is activated - see below) activated the phase follower automatically stops when a construction change marker (see also global settings chap. 1.2.1) or a change of the ensemble no. has been found. With the option **autostart** activated the phase follower automatically starts at the position of the nearest construction change marker to the left. The chosen mouse position is only shifted to the left and no time correction is done. With the option **last pick stop** activated the phase follower automatically stops at the position of the last manually set pick.

Auto pick: With this option activated the Auto-Pick window opens which allows you to set the autopick parameters and start the automatic picking (see also chap. 1.12.2.2). It is possible to bring this window to the front by pressing the right mouse button within the primary profile.

difference pick: With this option activated the travelttime difference between two successive picks is automatically determined. In addition the thickness is calculated based on this travelttime difference and the entered velocity (see PlotOptions chap. 1.7). First you must enter the reference pick and then the investigation pick. The two values travelttime distance and calculated thickness are plotted right to the investigation pick using the number font if the option show pickdifferences within the global settings menu (chap. 1.2) (also active when printing the profile). The number of total places and decimal places can be defined within the global settings (chap. 1.2.1) menu. The remove button as well as the sort and control buttons are disabled. After having activated this option all picks are reset.

With the option keep distance within the global settings menu (chap. 1.2) activated the x-position between the two picks are plotted in addition. With this option not activated the x-position of the second pick is automatically moved to the x-position of the first one.

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

use code: if activated the pick code is used for discriminating the individual picks:

- only the picks with the current code may be changed or removed
- different colors are used for the picks with different codes (with the option use layershow-col. activated the layershow colors are used)
- options correct, auto cor. and control only act on the picks with the current code

- when using the phase follower the option use code should be activated if several reflectors at the same distance are picked.

if deactivated the following holds true:

- all picks may be changed or removed
- the current pick color is used for all picks
- one correction type is used for all picks

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

use layershow-col.: With activated use code button the layershow colors are used for the pick display corresponding to their code. With deactivated use code button the picks will match the layershow colours based on the given layernumber when loading an existing pickfile.

load: This option allows to load new picks from disc. Please note that the current picks are lost with the loading of new picks. After selecting the option you can choose one or several datafiles from the file list. It is possible to load REFLEX formatted PCK files as well as different ASCII-formats:

ASCII-2-columns: each line of the ASCII-file corresponds to one pick and consists of two columns containing the distance and the time in the corresponding dimensions.

Tomo-coordinates: see chap. 1.12.2.1

- ASCII-file with each line containing the following values:

traveltime code x-shot y-shot z-shot x-receiver y-receiver z-receiver

or

traveltime code x-shot y-shot x-receiver y-receiver

The program automatically controls whether 3D- or 2D-data are present. If the 3D-format is used the picks are assigned to those traces which xyz-shot and receiver coordinates are next.

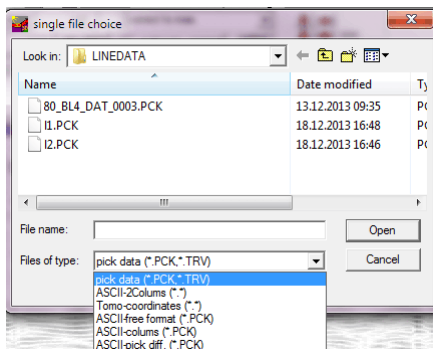
ASCII-free format: the program uses the last saved settings of the ASCII-free format (see chap. 1.12.2.1).

ASCII-columns: the program controls which options have been used when saving the picks using the ASCII-columns format (see chap. 1.12.2.1). The option acquisition times is not supported. If the picks had been saved using an older Reflexw version rows for the z-coordinates of the shots and the receivers must be included manually. In this case each line must contain the following informations:

distance offset(profileconstant) x-shot y-shot z-shot x-rec y-rec z-shot traveltime depth amplitude code

ASCII-pick diff.: the format corresponds to the format ASCII-Pick difference used for saving the picks (see also chap. 1.12.2.1).

To be considered: When loading the Reflex formatted PCK files the program automatically controls if the tracenumbers stored with the picks correspond to the



profile tracenumbers. If not a warning message window appears and a query if the pick tracenumbers or the pick distances shall be updated. The distances should be updated if after picking the traceincrement of the profile start has been changed. Otherwise the tracenumbers should be updated if the continuous picking or the phase follower shall be used afterwards as these functions use the tracenumbers for the identification of the picks.

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

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

refresh: allows to reload the actually loaded pick data and secondary picks. Possible unsaved changes will be lost.

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

undo: allows to undo the last pick action. This means for example that in continuous mode it would undo all of the picks made while the mouse button was held down. The option undo is only enabled if the small button on the left is activated.

autom.load: if activated the picks stored under the same filename like the currently loaded primary file (see also option automatic name under Pick Save MenuItem, chap. 1.12.2.1) are loaded automatically. It is possible to enter a postposition for the pickfilename which shall be automatically loaded.

sec.files: in combination with the option autom.load - if activated the picks with the same filename like the secondary files will be loaded automatically analogous to the option autom.load. The option sec.picks as well as the global settings option plot on 2.line will be deactivated automatically.

sort: sorts the picks with ascending order with respect to the distance axis. The continuous picking needs sorted data. Activate this option if the continuous picking is not working. The data are automatically sorted after having saved. If use code has been activated the sorting first will be done based on the code and then based on the tracenumbers.

x-flip: allows to flip the actual picks in x-(distance) direction.

velocity: enter the velocity which will be save with each pick and which may be used afterwards for a depth conversion.

correct: activate this option to correct the current picks. With the option use code activated only the picks with the current code are corrected for. With activated option **auto** located below the option correct the picks are automatically corrected during the setting. The following correct options are available:

- **correct to max.:** allows to correct the current picks to the extremum of the signal belonging to the pick.
- **correct to zero:** allows to correct the current picks to the zero crossing of the signal belonging to the pick. The zero crossing will not only be done to the nearest data point but to the interpolated values between the two datapoints of the zero crossing.
- **corr.max/min.time:** allows to correct the current picks to the extremum of the signal and the minimum time within a tracerange of a predefined number of traces (see input pickcorrect traces under global settings). Use this option for example to correct the picked cusp of a diffraction hyperbola.
- **corr.max/dist.range:** allows to correct the current picks to the extremum of the signal within a given tracerange of a predefined number of traces (see input pickcorrect traces under global settings). The timerange for searching the maximum is automatically restricted by the polarity of the current pick. Use this option for example to correct the picked cusp of a migrated diffraction.
- **correct for header elevations:** allows to correct the actual picks based on the traceheader elevations. Precondition is an activated plotoption correct header elevations.
- **set pick velocities:** stores the actual velocity to all picks.
- **refresh amplitudes:** takes the amplitude values for the picks from the actual file. The option might be useful if picking has been done based on another file and you want to take over these picks for another file including the actual amplitude values.
- **use correlation file:** The option allows the automatic correction of the picks based on a cross-correlation of the actual datafile with a reference file. The precondition is that the two files have the same number of samples, traces and the same timeincrement. The option might be useful for an automatic correction of first arrivals of different files which have been acquired at the same location but at different times. After having clicked on correct the reference file must be chosen. For reasons of accuracy, the cross-correlation is limited to a range of two signal periods (corresponding to the reciprocal of the dominant frequency) around the actual picks. Therefore, the correction is also limited to this range. The option is not available for the auto correction.

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

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

parameter `n.trace` defined within the additional `PickGroupBox` will be taken into account in order to decrease the number of picks.

If the option **all** is active all last picks which have at least one trace gap will be interpolated. The interpolation stops if two successive picks will be found.

The four **arrows** at the right side allow to move up and down and to the left and to the right the last set pick by one sample (trace) increment.

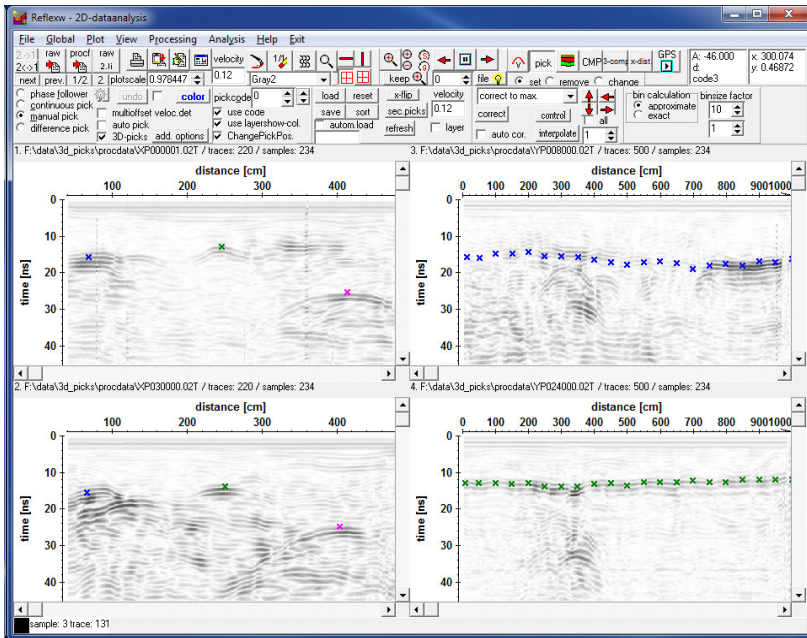
Activate the option **all** to move up and down (to the left or to the right) all picks by one or a multiple of the sample (trace) increment. The option **size** defines the amount (multiples of the actual increment) of the shift. With the option **use code** activated in addition only the picks with the current code are moved.

To consider when moving the picks to the left or the right: the xy-coordinates of the picks are not updated when using this option. This must be done during saving the picks with activated option xy-coordinates.

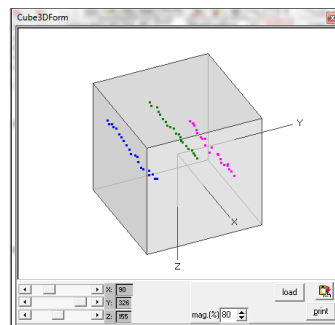
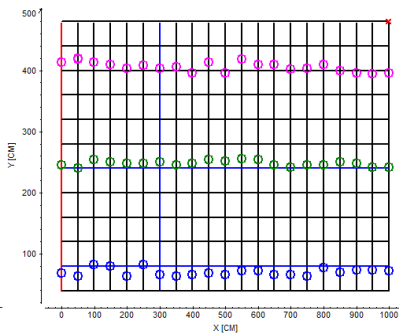
3D-picks: Activate this option if you want to pick within different individual (e.g. parallel) lines and you want as output one single pickfile. Precondition is that the traceheaders contain the coordinates. The option may be used for parallel and crossing lines as well as for GPS based crooked lines.

Picking may be done within all visible windows. One single pickfile will be generated for all used 2D-lines. The picking may be done for individual objects like hyperbola or for reflectors.

The actual picks will be displayed on all corresponding 2D-line (e.g. at crossing points).



The picks may also be displayed within the profile map using the option **view/profile line** and activated option **show act.picks**. A 3D-view is possible if the option **3D-pick cube** is activated in addition.



The options use code and use layershow-col. will be automatically activated as well as the option ChangePicPos. The pick save option check source/receivers positions will be deactivated.

With the option **reset 3D-picks** activated the 3D picks will be reset if a new file will be loaded except using the button next or prev.

For the display of the picks within the actual loaded 2D-lines a bin within the xy-plane must be defined in which the pick position falls.

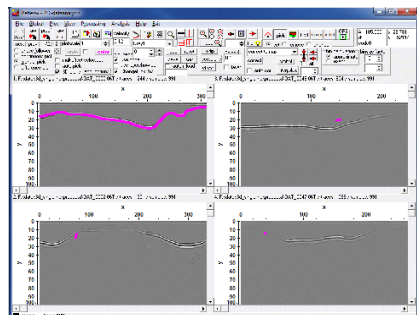
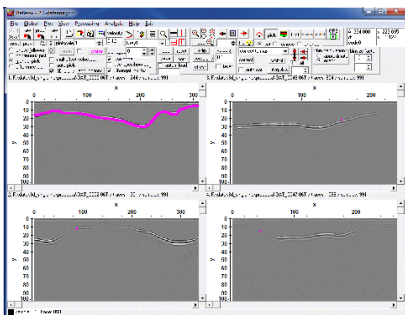
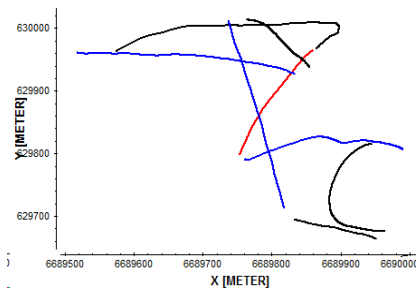
Two different **bin calculation methods** are available:

The fast method **approximate** controls if the pick lies on a straight line between the start and end-coordinate of the profile. A bin value (default 10) allows to increase/decrease the xy-range which controls if the pick belongs to the 2D-line or not. Increase this value if the pick will not be displayed (e.g. if gps-data are used which are not strictly straight) or decrease this value if the picks are displayed within more than one 2D-line.

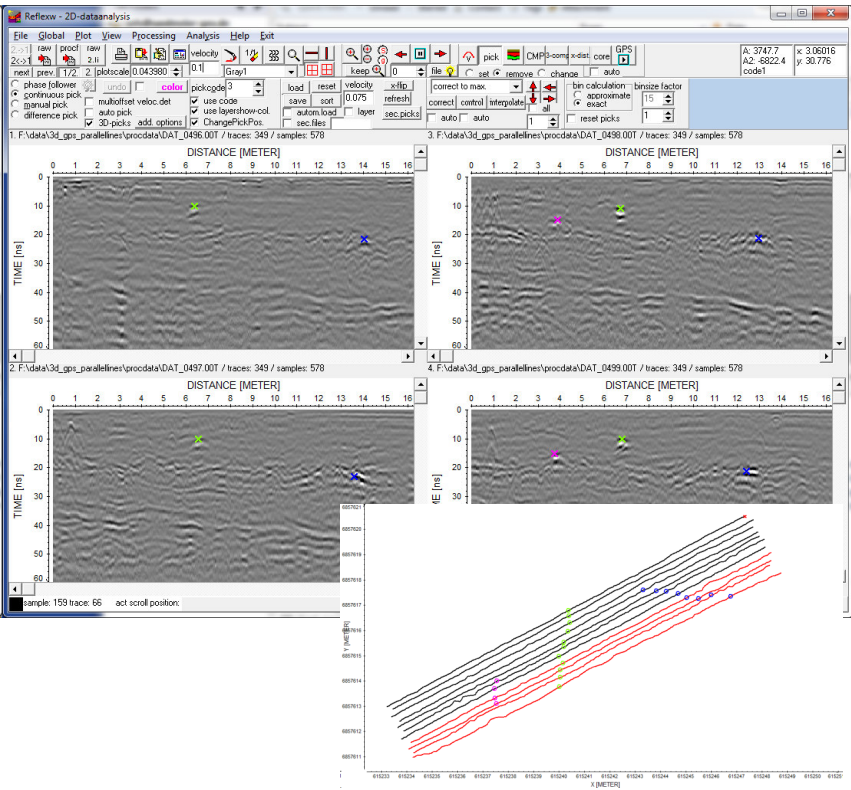
The slow method **exact** controls if the pick coordinates and the traceheader coordinates match. Again a binsize factor is used in order to increase the bin value (default 1 - means that the matching is true if the range is smaller than the actual traceincrement).

The bin calculation approximate provides worse results but the computation time is much less whereby it may serve for a first fast check. Depending on the irregularity of profile lines the binsize factor must be increased accordingly. During picking in case of the bin calculation exact the picks will only be displayed within the actual window in order to speed up the picking process. For a correct display of the picks in all windows the data must be replotted.

Example 1: pick a reflection within crooked crossing lines. The four lines (one red and three blue) are loaded within the four windows. The picking is done within the primary window. If the bin.calculation approximate is activated the crossing picks are displayed automatically (right lower picture). As this method is not very precise (he profiles do not ly on a more or less straight line) you should replot the data after picking with the bin method exact activated (left lower picture).



Example 2: pick different linear elements visible as hyperbolas within the 2D-lines. The pick codes serve for a discrimination of the individual objects, e.g. pipes. The bin method exact has been used because of the irregularity of the 2D-lines. During picking only the picks on the actual line will be displayed. As no crossing lines are present this is no restriction. Even if there are a few crossing lines it does not make sense to use the approximate method but the exact method should be used and by replotting all lines possible crossing points will be displayed within the crossing lines.



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

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

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

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

load perpendicular picks: allows to load and to correct existing picks for the xy-coordinates stored within the traceheader of the current datafile (see also trace header Edit menu, chap. 1.9.6). Those picks which xy-coordinates do not correspond to any existing xy-coordinate of the current datafile are automatically cancelled. The option might be useful if existing picks are loaded into a crossing line. The parameter **corr.xy-coord bin(%)** under global settings (chap. 1.2.1) controls the size of the crossing bin. Increase this parameter if the option control xy-coord does not generate any picks.

remove non double picks: allows to remove all picks for which only 1 pick for one distance position exists. The option is useful for a continuous picking of a layer thickness. For that purpose first pick the upper border of the layer and then the lower border. Then use the option remove non double picks in order to ensure to have 2 picked values for each location. Then you may use the ASCII-pick difference output within the Pick Save MenuItem (chap. 1.12.2.1) in order to output the thickness of the picked layer.

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

The screenshot shows the 'PickGroupBox' dialog. On the left, there are input fields for 'n.trace' (set to 1) and 'max.diff' (set to 5). Below these are checkboxes for 'autostop', 'autostart', and 'last pick stop', and a 'delay' spinner set to 1. There are buttons for 'remove non double picks', 'load perpendicular picks', and 'marker picks'. On the right, there's a 'pick code' section with buttons 'change pick code', 'set pick code to all', and 'set pick code to range'. Below that are 'min.dist' and 'max.dist' fields (both 0), and checkboxes for 'show pick code', 'code transparent', and 'code auto-increase'. At the bottom right is a 'move based on sec. picks' button. At the bottom left, there's a 'sec. picks' section with a 'color' button and checkboxes for 'only sec. lines' and 'use layershow-col.'. At the bottom right, there's a 'not active picks' section with a 'color' button.

set pick code to all: allows to overwrite the codes of all picks with the current code.

set pick code to range: allows to set the actual code on all picks lying within the distance range defined below (options **min.dist.** and **max.dist.**).

move based on sec. picks: Activating the option moves the current picks to larger times based on the loaded secondary picks. The option might be used for example for an adaptation of existing pick files to a topographic correction. For this purpose the interpretation flow is the following:

- a. pick the reflections based on the non topographic corrected profile
- b. perform the topographic correction
- c. pick the first arrival within the topographic corrected profile
- d. load these first arrival picks as secondary picks
- e. load the wanted pick file and activate the option "move based on sec. picks" - the picks will be adapted to the topographic correction.

To be considered: Moving is done based on the times of the sec. picks - therefore the starttime of the profile should be 0. If this does not hold true or if you are not able to pick the very first arrival you still have the possibility to move the surface reflection picks up or down using the red arrows with activated option "all".

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

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

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

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

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

change pick-pos.: by default the distance position of a pick will not be changed when using the change option (changing existing picks). With change pick-pos. activated it is also possible to change the distance position of existing picks.

The option **multioffset veloc.det.** allows a fast determination of the 2-dimensional velocity distribution from multioffset data.

Precondition are Constant Offset lines with the source-received distance stored within the fileheader (option S/R-distance). A Zero Offset profile must be present or it must be created from a Constant Offset line using the processing option dynamic correction. All multioffset lines must be corrected for the start time.

The following options are available:

multioffset files: Choose the multioffset files from which the velocities shall be determined

min. vel./max. vel.: defines the velocity range

average: defines the range in the given distance dimension for an averaging

median: if activated the median velocity within the average range is determined - otherwise the mean velocity

t-shift: enter a time shift if the data had not been corrected for the picked phase. In the following example time zero has been chosen for the extremum of the first positive phase and the reflection (which has a positive reflection coefficient) has also been picked for the positive (white) phase. Therefore a time shift is not necessary.

long offset approx. If activated a fourth-order moveout equation is used instead of the normal vrms equation.

$$t(x) = \sqrt{(t_0^2 + x^2 / V_{ave}^2) / (1 + \frac{gx^2}{t_0^2 V_{ave}^2 (g+1)})}$$

$$g = (V_{rms} / V_{ave})^2 - 1$$

The vertical heterogeneity must be entered manually. The parameter **heter.(%)** defines the parameter g in percent. It is possible to get an estimate of g within the CMP-velocity analysis by crating a manual 1D-model and to use the option long offset approx. there. With the option view/model spreadsheet the vertical heterogeneity for the different layers of the actual model will be displayed and may be used here for the velocity determination. The option is useful if a strong vertical heterogeneity of the velocities is given and for long offsets.

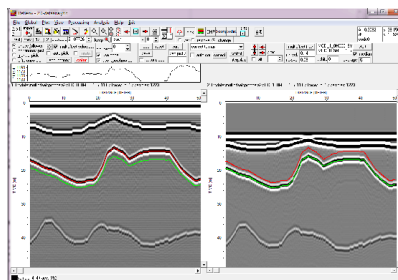
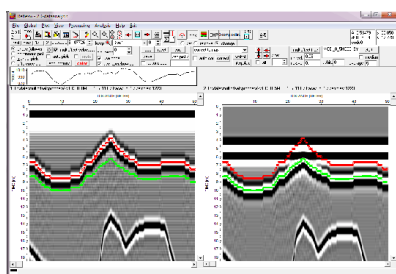
Start: starts the automatic velocity distribution. Dependent on the picked interface within the Zero Offset lines the program automatically detects the reflector within the other multioffsets lines. For this purpose each pick will be shifted based on the entered velocity range and the S/R distance and the best fit (identical polarity, highest amplitude value) gives the mean velocity for this pick. A control of the method is given if one of the multioffset files has been loaded as a secondary file and the option sec.picks has been activated. Then the found picked

interface will be plotted in addition (lines in green).

The calculated mean velocities will be saved with the picks and are shown in an additional window (option view/add. 2. Column data will be automatically activated).

The following pictures show the result for two reflectors.
remark: for the 2. layer the black (negative) phase has been picked because of a phase shift of this reflection due to a negative velocity contrast (high to low velocities).

Activating the option **interpolate** allows to interpolate the calculated velocities within the additional column window. After having activated the option click on the first velocity and then on the second one. Between these two velocities a linear interpolation will be done. The option may serve for a smoothing of the derived velocities. Precondition for the option is that the column window has not been overlaid (option overlay primary section must be deactivated) and manual scaling has been activated..



1.12.2.1 Pick save MenuItem

Save the current picks using different formats.

Control Panel section:

filename: specify the filename. Based on the chosen formats the file will have different extensions and will be stored under different directories:

ReflexWin: extension PCK and storing under the path LINEDATA.

ASCII-free format, ASCII-columns and ASCII-pick difference: extension PCK and storing under the path ASCII

ASCII-kml file: extension kml under the path ASCII

ASCII-tomography: extension TOM and storing under the path ASCII

GeoTomCG: extension 3DD under the path ASCII

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

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

automatic save: Activating this option allows you to automatically save the actual picks using the Reflex Win format and/or the ASCII-columns format if a new profile will be loaded or if the pick option will be left. The automatic filenames will be activated - the pickfilename is automatically determined from the filename of the current profile. The actual settings of the ASCII-columns format will be used. The option export several existing picks into 1 ASCII-file for the ASCII-columns format is not supported and will be automatically disabled if activated.

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

close after save: if activated the pick save menu will be automatically closed after saving the picks.

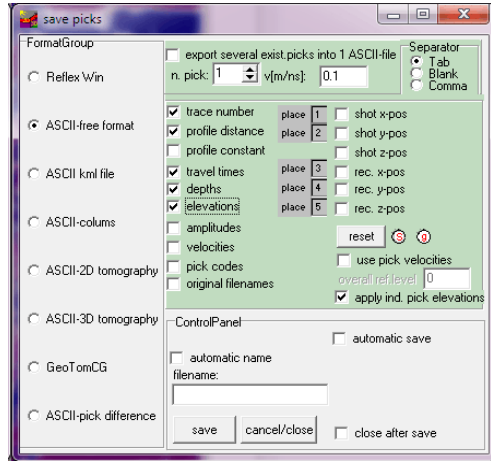
cancel/close: close the window without saving the picks or simply close the window if the picks have been saved before.

FormatGroup section:

The following output formats are supported:

ReflexWin: new binary ReflexW pick format now using double precision (64 bit) for the xy-coordinates - data cannot be loaded with older ReflexW version (up to ver. 5.0.8).

Ascii-free format: The format allows to save the picks on an ASCII-file using free positioning and separator. Each line contains the desired parameters for one pick in the order (place) the parameters have been chosen. The option reset resets the choice. The small buttons “s” and “g” allow to save and load the actual settings on the file FreeAsciiFormat.txt stored under the program directory. This saved setting is also used if the picks shall be reloaded again. With n.pick greater 1 it is possible to restrict the number of picks to be written out.



Three different separator characters can be chosen (Tab, Comma or Blank).

With the option use pick velocities activated the velocities stored with the picks are used in order to transform the traveltimes into depths. Otherwise the velocity given at the top is used. In case of already time depth converted picks the velocity will be automatically set to 2 which is identical to a transformation factor of 1.

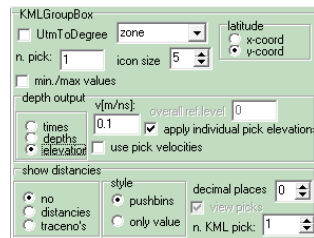
With the option apply ind. pick elevations activated the elevations of the picks are determined from the difference of the pick z-receiver coordinates and the picked traveltimes converted in depths. Otherwise the overall ref.level will be used in order to determine the elevations from the depths.

With the option export several existing picks into 1 ASCII-file activated you may export several existing pickfiles into 1 ASCII-file. In this case you should output the shot and receiver xy(z)-coordinates and not the profile constant.

ASCII kml file: allows to save the picks using the Google kml format.

The following options must be entered:

UTMTToDegree: activate this option if the xy-coordinates are not given in degrees but in UTM meter coordinates. Then you must enter the correct utm-zone and the xy-coordinates will be automatically transformed into degrees. The groupbox **latitude** defines where the latitude values have been stored (either on x- or y-coordinates).

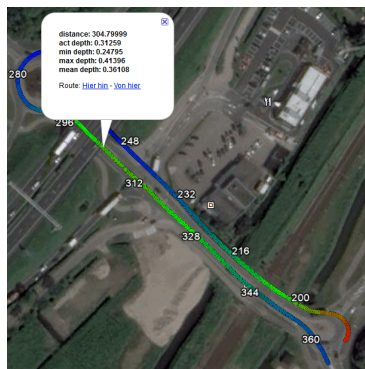


The **icon size** defines the size of the icons. The option **n. pick** allows to export only each n. pick.

If **depths** has been activated the traveltimes will be transformed into depths using the velocity listed on the right. In case of already time depth converted picks the velocity will be automatically set to 2 which is identical to a transformation factor of 1. With the option **use pick velocities** the velocity stored with each pick will be used for the time-depth conversion. If **elevations** has been activated the traveltimes will be transformed into elevations. With the option apply ind.pick elevations activated the elevations of the picks are determined from the difference of the pick z-receiver coordinates and the picked traveltimes converted in depths. Otherwise the overall ref.level will be used in order to determine the elevations from the depths.

The groupbox **show distances** allows to show the **distancies** or the **traceno's** along the line stored with the picks. If **pushbins** is activated a yellow pushbin will be displayed in addition, if only value is activated only the distancies/traceno's will be shown. The option decimal places is used for the display of the distancies. With the option show picks activated the picks will be displayed in addition - if deactivated only the distancies will be displayed. The option n.KML pick allows to show only n. distance of the exported picks (the total number of distancies therefore depends both on n. pick and on n. KML pick).

Each datapoint within the KML-file contains an information box about the distance, actual, min., mean and max. depth. A red-green-blue color scheme for the icons will be used with red corresponding to the smallest values and blue to the largest depth values.



DXF: allows to save the picks using the DXF format. The picks may be exported either as **point**, **line** or **circle**. The x and y-coordinates are exported as well as the traveltime or converted depth (elevation). With the option **use pickcode** activated the **layer** group code within the dxf-file will be set to the pickcode which allows an easy discrimination of the picks also for the line type. For the circle type the **radius** must be entered. The DXF export is also very suitable to 3D-picks.

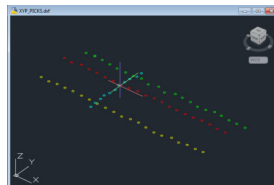
The following options may be entered:

An optional **utm conversion** is possible. The groupbox **latitude** defines where the latitude values have been stored (either on x- or y-coordinates).

With the option **InterchangeXY** activated the xy-coordinates will be interchanged for the export. This might be useful for example if the latitude has been stored on the x-coordinate and no utm conversion shall be done.

The option **n. pick** allows to export only each n. pick.

If **depths** has been activated the traveltimes will be transformed into depths using the velocity listed on the right. In case of already time depth converted picks the velocity will be automatically set to 2 which is identical to a transformation factor of 1. With the option **use pick velocities** the velocity stored with each pick will be used for the time-depth conversion.



If **elevations** has been activated the traveltimes will be transformed into elevations. With the option apply ind.pick elevations activated the elevations of the picks are determined from the difference of the pick z-receiver coordinates and the picked traveltimes converted in depths. Otherwise the overall ref.level will be used in order to determine the elevations from the depths.

ASCII-columns: convert picks to an ASCII file whereby every pick uses one line. With the option **export several existing picks into 1 ASCII-file activated** you may export several existing pickfiles into 1 ASCII-file. Each line of the ASCII-file contains the following values:

traveltime(12:6), depth(10:4), amplitude(yy:xx), x-shot(yy:xx), y-shot(yy:xx), z-shot(yy:xx), x-receiver(yy:xx), y-receiver(yy:xx), z-receiver(yy:xx) where yy defines the number of places and xx defines the number of decimal places given within the global settings menu (chap. 1.2.1). The x-,y-,z-shot and receiver coordinates have been taken from the traceheaders of the individual profiles when picking the onsets (see also trace header Edit menu - chap. 1.9.6). Depths and amplitudes are set to zero if the corresponding options depth and amplitudes are deactivated.

With the option **export several existing picks into 1 ASCII-file deactivated** it is possible to save the existing picks into an ASCII-file. Every pick uses one line. Each line contains the following values: coordinate in profiledirection(10:3), coordinate in profileconstant(10:3), x-traceheadercoordinate (optional,yy:xx), y-traceheadercoordinate (optional,yy:xx), z-traceheadercoordinate (optional,yy:xx), travel time(12:6), depth(10:4), amplitude(yy:xx) where xx defines the number of decimal places given within the global settings menu (chap. 1.2.1). The number of total places yy can also be defined within the global settings menu. By default the travel times and also the plane-coordinates of the picks are written on a file. Additionally the travel time can be converted into depths (option **depth** activated) on the basis of a specified constant velocity (In case of already time depth converted picks the velocity will be automatically set to 2 which is identical to a transformation factor of 1) or on the velocities stored with the picks (Option **use pick velocities** activated) and the current amplitude of each pick can be saved (option **amplitudes** activated). If no depths and/or amplitudes shall be extracted, these values are set to zero. Optionally the xyz-coordinates of the receivers (specification **xyz-rec.coordinates**) stored in each trace header may be written out (see also trace header Edit menu - chap. 1.9.6). This allows you for example to write out non equidistant xyz-coordinates or the xy-coordinates, if the profile is not orientated into one direction (x or y). The xyz-coordinates are only saved on file if chosen. With the option **xyz-shot/rec.coord.** activated both the shot and the receiver xyz-coordinates are written out. With the option **add meridian stripe** activated the entered meridian stripe value will be added at the first place of the easting value. Optionally the acquisition times of each trace stored within the traceheader is written out (option **acquisition times** activated). Optionally the pickcode of each pick can be written out (option **pickcodes** activated).

The parameter **n.pick** controls the pick number to be output: only every n. pick will be output. The file has the extension PCK and will be stored under the path ASCII under the current projectpath.

With the option **original datafilename** activated the original datafilename (without path) is output in addition within the first column. With the options **trace number** and **sample number** activated the tracenummer and the samplenummer are also exported at the beginning of each line of the ASCII-file.

Activating the option **profileconst.->1.column** allows to write out the profileconstant within the 1. column and the profiledirection within the 2. column. The option might be useful for the export of picks of crossing profiles.

If picks have been loaded within the 3-component analysis the actual **angles** can be exported together with the picks when using the ASCII-columns format (option angles activated).

Ascii-2D tomography: every pick uses one line. It is possible to save the picked **traveltimes** or the **amplitudes**. Each line contains the following values: `traveltime(yy:xx)` or `amplitude, code(10:0)`, x-traceheadercoordinate of the transmitter, y-traceheadercoordinate of the transmitter(`yy:xx`), x-traceheadercoordinate of the receiver(`yy:xx`), y-traceheadercoordinate of the receiver(`yy:xx`) where xx defines the number of decimal places given within the global settings menu (chap. 1.2.1). The number of total places yy can also be defined within the global settings menu. The xy-coordinates are taken from the corresponding traceheaders of each trace (see also trace header Edit menu - chap. 1.9.6) during picking. The file has the extension TOM and will be stored under the path ASCII under the current projectpath. The format corresponds to that format needed by the 2D-tomographic `traveltime-inversion` and `amplitude-inversion`. The `distancedimension` is always METER, the `timedimension` is either ms or ns. If the `timedimension` of the original data is μ s, the `traveltimes` are automatically transformed into ms-data for storing.

Ascii-3D tomography: every pick uses one line. Each line contains the following values: `traveltime(yy:xx)`, `code(10:0)`, x-traceheadercoordinate of the transmitter, y-traceheadercoordinate of the transmitter(`yy:xx`), z-traceheadercoordinate of the transmitter(`yy:xx`), x-traceheadercoordinate of the receiver(`yy:xx`), y-traceheadercoordinate of the receiver(`yy:xx`), z-traceheadercoordinate of the receiver(`yy:xx`) where xx defines the number of decimal places given within the global settings menu (chap. 1.2.1). The number of total places yy can also be defined within the global settings menu. The xyz-coordinates are taken from the corresponding traceheaders of each trace (see also trace header Edit menu - chap. 1.9.6) during picking. The file has the extension TOM and will be stored under the path ASCII under the current projectpath. The format corresponds to that format needed by the 3D-tomographic `traveltime-inversion`. The `distancedimension` is always METER, the `timedimension` is either ms or ns. If the `timedimension` of the original data is μ s, the `traveltimes` are automatically transformed into ms-data for storing.

GeoTomoCG: this format should be used when the picked data are interpreted using the tomography program GeoTomoCG.

2 comment lines are at the beginning. Then every pick uses one line. Each line contains the following values: pick number (`5:0`), x-traceheadercoordinate of the transmitter (`yy:xx`), y-traceheadercoordinate of the transmitter(`yy:xx`),

z-traceheadercoordinate of the transmitter(yy:xx), x-traceheadercoordinate of the receiver(yy:xx), y-traceheadercoordinate of the receiver(yy:xx), z-traceheadercoordinate of the receiver(yy:xx), travelttime(yy:xx) where xx defines the number of decimal places given within the global settings menu (chap. 1.2.1). The number of total places yy can also be defined within the global settings menu. The xyz-coordinates are taken from the corresponding traceheaders of each trace (see also trace header Edit menu - chap. 1.9.6 and CMP-processing geometry, chap. 1.12.4.1) during picking. The file has the extension 3DD and will be stored under the path ASCII under the current projectpath. Activating the option **store y on z-coord** allows you to store the y-traceheader-coordinates on the z-coordinates of the GeoTomoCG-file. The y-coordinates of the GeoTomoCG-file will be set to 0. This option may be useful for a 2D-tomography in the x-z plane because the definition of the traceheader coordinates within REFLEXW are much more easier for the xy-plane than for the xz-plane (see CMP-processing geometry, chap. 1.12.4.1).

ASCII-pick difference: allows to output the travelttime difference of two successive picks together with a calculated thickness. The picks may not be located at the same distance position. The pick pair is defined from the ascending order in distance direction. Therefore if you are using the manual or continuous pick for defining a layer thickness you should use the option remove non double picks (see Pick MenuItem, chap. 1.12.2) in order to ensure that the correct two picks are always used for calculating the pick difference. Every pick pair uses one line. Each line contains the following values: distance for pick 2, distance difference, thickness, travelttime difference, travelttime 1, travelttime 2, Pick Code. The number of total places and decimal places can be defined within the global settings (chap. 1.2.1) menu. The thickness is calculated from the travelttime difference and the entered velocity. With the option **xy-coordinates** activated the xy-shot and receiver coordinates of the second pick are written out in addition. The xy-coordinates are taken from the coordinates stored in each trace header (see also trace header Edit menu - chap. 1.9.6) of the current profile. By default the picks are stored under ASCII under the current projectdirectory. Using the option **path** it is possible to enter another path. Then the filemenu opens. If no filename has been specefied before you must enter such a name. To be considered: it is still necessary to activate the option save in order to save the picks.

ReflexWin section:

global code: A code is queried which serves for differentiating picks of different phases from one data set. Like this signal arrivals of compression waves can be marked with a P and the ones of shear waves with a S. This code is used within the LayerShow MenuItem (chap. 1.12.3). If no global code is entered no coding is shown within the layershow.

mean or layer velocity: Additionally a velocity can be preset which is used for the conversion of the travel times in depths under the option layershow. You have to specify a mean velocity for the total overburden or a layer velocity. Within the layershow you may specify whether mean or layer velocities are used for the time-depth convesion (see chap. 1.12.3.1).

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

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

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

old format (up to ver. 7.5): allows to store the picked data using the pck-format which has been used by older Reflexw versions (up to version 7.5).

1.12.2.2 Auto-Pick MenuItem

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

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

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

At present it is possible to pick the following objects (option **object type**):
single objects like small diffractions (e.g. from rebars) - data should be migrated (see also chap. 1.11.9)

first arrivals (e.g. from refraction seismics)

continuous reflector

phase follower

The following parameters are valid for all object types:

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

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

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

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

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

slope °: enter the slope angle in degrees for determining the time/distance range for the automatic search of the picks. With a value $\neq 0$ the start and end time for the detection are changing with distance ($\text{distance} \cdot \tan(\text{slope})$). A value $\neq 0$ allows you for example to automatically detect a continuously dipping reflector.

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

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

CorrectBox: within this box you may determine how the picks are corrected after having detected:

no: no correction is done.

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

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

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

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

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

Object type **single objects:**

A single object is determined if the following criteria are fulfilled:

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

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

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

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

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

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

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

Object type **first arrivals:**

A first arrival is determined if the following criteria are fulfilled:

1. The polarity must fit.
2. The onset must be located within the given time/distance range.
3. The amplitude must exceed a given threshold.
4. The instantaneous frequency must be in the range of the mean instantaneous frequency. The range is given by the mean instantaneous frequency and the frequency scale parameter.
5. The timeshift between the actual pick and the median value does not exceed a given value.

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

amp.scale: enter a scaling factor for determining the amplitude threshold. The program automatically extracts the max. amplitude for each trace within the given time range and at the given polarity. If the option **use overall max. value** is activated the overall maximum value for all traces within the given time range is used instead of the max. value for each trace. In both cases the threshold is determined from this max. value multiplied by the amp.scale factor.

frequency scale: enter a scaling factor for determining the range of the allowed instantaneous frequencies. A factor of 1 means that a 100 % deviation from the mean instantaneous frequency is allowed. Accordingly a factor of 0.1 means 10 % deviation and a factor of 10 1000 % deviation. The program automatically calculates the mean instantaneous frequency for all traces within the given time/distance range. The threshold range is determined from this mean value multiplied by the frequency scale factor.

max. timediff and search size: the program controls whether the timeshift between the actual pick and the median value within the given search size (symmetrical around the pick position) does not exceed a given value (option max.timediff).

Object type **continuous reflector:**

A continuous reflector is automatically picked at those positions where:

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

The program controls whether the timeshift between the actual pick and the median value within a given **search size** (symmetrical around the pick position) does not exceed a given value (option **max.timediff**). If this is the case the value will be replaced by the median value. With the option **up down tendency** activated the program controls whether a tendency toward downward or upward times is given. If such a tendency has been evaluated no replacement of the actual pick will be done except the timeshift exceeds 4 times the given max. timediff value.

With the option **control phase** activated the program checks whether an onset of the same polarity within a **search timerange** (should be about the signal period) exists at smaller times with a minimum of 30 % of the amplitude of the chosen pick. If this holds true the chosen pick will be replaced by the shifted one. The option is useful if the signal normally exhibits two etrema of the same polarity.

Object type **phase follower**:

A continuous reflector is automatically picked using the phase follower. The start of the phase follower will be done automatically by defining the **start/end time** and the **polarity**. The program automatically searches the max. value within this time windows of the given polarity at the start position. The **distance range** may be total or restricted to a start and end distance.

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

The parameters within the control panel box are the following:

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

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

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

batch start: start the autopicking for a choosabele number of files. After having activated this option you must select the wanted profiles from the filemenu. Afterwards the automatic picking is started for all chosen profiles. The detected picks of the different profiles are stored on individual files using the ReflexWin pickformat (extension PCK under the path LINEDATA) and the filename automatically determined from the acutal profile filename (see also option automatic name under Pick Save MenuItem, chap. 1.12.2.1).

close: close the auto-pick window.

1.12.3 LayerShow MenuItem

This option offers the possibility to combine individual pick files stored on file (see option PICK), to plot them together with the wiggle-files and to output them in report form on printer or file. The maximum number of combined picks per layer boundary is equal to the max. number of traces per profile. The maximum number of samples per distance point and thus of layer boundaries is 100. The layer number within each pickfile serves to identify the individual picks. At each distance point the layer depths must increase with ascending layer number. Therefore you must choose the layer number when saving the picks in such a way that the correct layer order will be ensured especially when using a velocity file for creating the layershow and when output the thicknesses of the individual layers.

After selecting the option you have the choice between

create - i.e. to create a new layer show from pick files and save them on file

load - load an existing layer show

create report - i.e. to create a reportfile based on the current layer show.

Create velocities - i.e. to create layer velocities for the individual layers either based by manual input or based on an amplitude calibration

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

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

The option **view 1.-4.** allows to view up to 4 different layershows. The scaling (x and z) is the same for all 4 layershows. No zooming or scrolling is possible. After having pressed the replot or reset button the original data will be displayed.

The option **2.line/lay** allows to load a second line together with a layershow. Accordingly to the possibility of load up to 4 different profiles the display can be splitted into pure vertical split, pure horizontal split or both vertically and horizontally (hor./ver.split or ver./hor.split activated).

If the option **show picks** is activated, additionally in the upper window the combined travel times (not depth-converted) are plotted as continuous layer boundaries, so that a direct comparison between raw data and picked arrivals is possible.

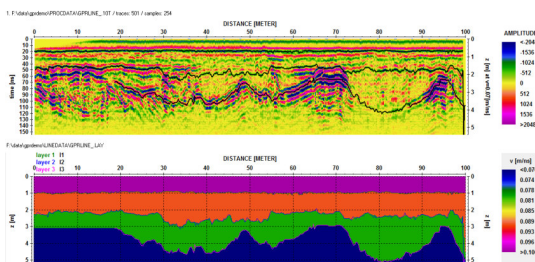
With the option **dist.axis** activated the distance axis is plotted for the lower layer window.

With the option **grid** activated a grid is plotted within the lower layer window.

With the option **fill layers** activated the layer above the boundary is filled using the current layershow color.

The option is also available for printing. The color for the lowermost layer underneath the lowermost boundary is always white. To be considered: if the boundary is not continuous the color of the lower boundary will be used. With the options **with velocities** activated in addition the resulting

layershow will be filled using the pick mean or layer velocities. Two different color bars are shown if the color bar view is active. The velocities below the lowermost layer are not defined. With the option **default velocities f. last layer** deactivated the velocities above the layer boundary are used also for the region below the boundary of the lowermost layer. With the option **default velocities f. last layer** activated the default velocity is used for the region below the layer boundary.



The current codings of the individual boundaries (see global code within the Pick save Menuitem, chap. 1.12.2.1) are displayed based on the settings within the **legend** box using the symbol font. The colors of the codings are automatically determined from the layer numbers. The option **legend name** allows you to define the default prefixname of the legend - if this name is set to nil no legend is plotted. The size of one coding is calculated from the option **size** (enter the size of one coding in number of characters). The option **rows** defines the number of rows the legend is subdivided into. You have the following possibilities for defining the **legend position**:

Above means plotting the codings in the free space between the two windows.

Upper left means plotting in the upper left corner of the layer window.

Lower left means plotting in the lower left corner of the layer window.

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

The option **export** allows the export of the layershow into

- a. an **ASCII modelfile** - this file can be imported within the modelling module using the option file/load from ASCII. To be considered for GPR-data: the wave type must be set to electromagnetic before importing the ASCII-file. Then the velocities within the ASCII-file are automatically converted into epsilon values.
- b. a **Reflexw rasterfile** containing the layershow velocities if the option **generate Reflexw rasterfile** has been activated. The size of the rasterfile is controlled by the options x-points and y-points. By default the values are set to the sample and scan number of the original file. With the option **horizontal extrapolation** activated all layers are extrapolated to the start and end of the model. Gaps inbetween a layer boundary are always closed by a linear interpolation. The rasterfile will be stored under the directory rohdata. If a rasterfile containing the epsilon values is wanted the option conv. v to epsilon under declipping/arithmetic functions allows to convert the velocities into epsilon values. The rasterfile may

also be used for example in order to do a timedepth conversion. In this case the layershow should have been created using layer velocities and for 2D-velocity file rasterfile layer should be used.

The velocities below the lowermost layer are not defined. With the option **default velocities f. last layer** deactivated the velocities above the layer boundary are used also for the region below the boundary of the lowermost layer. With the option default velocities f. last layer activated the default velocity is used for the region below the layer boundary.

If a layer pinching out has been picked you should activate the option **pinching out layer** because otherwise this layer will be assumed to be continuous.

1.12.3.1 Create LayerShow

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

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

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

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

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

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

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

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

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

Choosing "**layer single vel.**" means that the velocity specified at the saving of the picks of the corresponding layer (see also SAVE under PICK) is assumed to be a layer velocity and it is used for the time-depth conversion. Only one value for the complete layer is possible. The velocity is given as the layer

velocity. Therefore the time-depth conversion of each layer boundary depends on the conversion of the upper boundary (see also option interpolate layer). The result is the same as using the option velocity file with a constant velocity for each layer (see below).

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

Choosing "**velocity file**" means that the layer velocities given in an ASCII file (for the format description see option "create Layershow velocity", chap. 1.12.3.3) are used for the time-depth conversion. The wanted velocity file must be chosen from the file menu. The velocities for each layer may laterally vary and they are normally given as layer velocities except if the second line within the ASCII-file includes the string "mean velocities". The velocities given in the file are defined as the velocities above the layer boundary. The time-depth conversion of each layer boundary depends on the conversion of the upper boundary (see also option interpolate layer). If no values are defined at the given position, an interpolation between the two next given values is used. Assumption: the layers are numbered subsequently starting at 1 at the uppermost layer boundary (normally the air-surface boundary).

example: three different layer boundaries are defined at one position with $t_1=10\text{ns}$, $t_2=15\text{ns}$, $t_3=22\text{ns}$. Choosing "mean pick velocity" means that the given mean velocities of $v_1=0.1\text{ m/ns}$, $v_2=0.09\text{ m/ns}$ and $v_3=0.085\text{ m/ns}$ are used. This results in the following depths: depth1: 0.5 m, depth2: 0.675 m, depth3: 0.935 m. Choosing "velocity file" means that the given layer velocities at the distinct position are read from an ASCII file: e.g. $v_{lay1}=0.098\text{ m/ns}$, $v_{lay2}=0.085\text{ m/ns}$ and $v_{lay3}=0.070\text{ m/ns}$. This results in the following depths: depth1: 0.49 m, depth2: 0.7025 m and depth3: 0.9475 m.

interpolate layer: option is used if the velocities are read from an ASCII-file (see option VELOCITIES/velocity file) or if the option layer pick velocity has been chosen. The option controls how the time-depth conversion for a special layer is done if any upper layer is not continuously picked.

If the option is TRUE every upper layer is assumed to be continuous even if it is not picked. In this case either an interpolation or an extrapolation both for the layer points and the velocities is done. Based on these calculated values the time-depth conversion of the current layer is done. Activating this option is useful if you may assume that the upper layer is continuous but hardly to interpret in the reflexion data. The interpolation or extrapolation is done over the complete distance range except the option **< comments >** is activated. If this option is activated an upper layer will only be interpolated or extrapolated within those comment marker distance ranges where it really exists. It will be neglected within those comment marker distance ranges where it has not been picked.

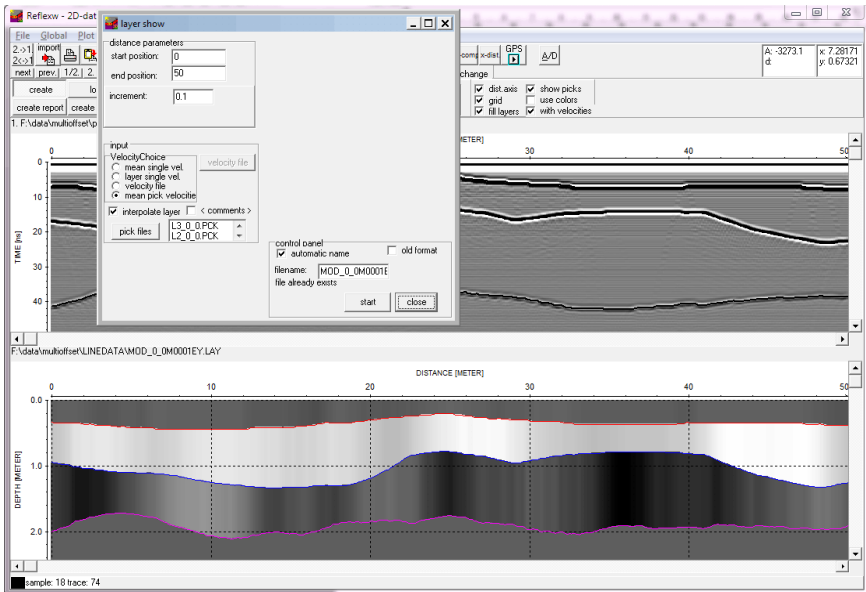
If the option is deactivated, the velocities of the next upper picked layer are used. This might result in a sharp step of the depth of the current layer at the point where any of the upper layers is broken. Deactivating this option is useful for the case of a vanishing upper layer.

< comments >: if activated the interpolation or extrapolation of an upper layer is only done within those comment marker distance ranges where the upper layer

really exists. The option is useful for example if the comment markers define construction changes with different layering. A comment marker distance range is defined between two successive comment markers and from the profilestart to the first marker and from the last marker to the end of the profile. If no comment markers are present the whole distance range of the profile will be used. If deactivated the inter- and extrapolation is done over the complete distance range of the profile.

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

close: terminates the window.



1.12.3.2 Create LayerShow report

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

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

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

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

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

start position, end position, increment, average: specify the start and end coordinate in the set dimension. The increment and average must be a multiple of the layershow increment. This will be automatically controlled. Average defines the range over which all values are averaged. By default average is set to increment. If average is chosen larger than the distance between two picks within the layershow file, an averaging is performed over the corresponding number of data points.

only construction data: if activated only picks at the construction change markers (see also global settings, chap. 1.2.1) are used. The number of reported values is equal to the number of construction change markers. Averaging is done like for the normal report output.

summary comment data: option replaces the old option "summary construction data". If activated all picks between two comment markers are averaged. With the option **only construction data** activate only those markers containing the construction change marker text will be considered. The summing up can be chosen from average, median and mode to be found under **summary over**. In addition the comment will be output in the first column of the report. The start position will be rezeroed at a new comment position if the option rezero at comment is activated (exception: a construction change marker (see also global settings, chap. 1.2.1) has been found). The number of reported values is equal to the number of comment markers plus 1 if the complete distance range will be covered. The total start and end position can be entered which allows to restrict the distance range of the report output. The option is useful if you only want to

pick a single point to represent material depth between two construction changes.

To consider: the number of characters within one report column is restricted by the option number of digits. Therefore you must adapt this number to the max. size of the comments if you do not want to lose any information of the comments.

rezero at comment: if activated the start position will be rezeroed at the new comment position except a construction change has been found.

check single values: option enabled for summary comment data output. With the option activated a check will be included if only one pick exists at the start and/or at the end of the distance range defined by two successive comment markers. Those picks will be neglected in this case. This might be useful if due to rounding problems a pick reaches into a non desired comment marker range.

calculate thicknesses from average depths: controls how the thicknesses are calculated for summary comment data. If activated the thicknesses are calculated from the averaged depths. Activate this option only if the layers are quite continuous within each average area defined by the comment data.

reverse chainage: if activated the chainage is reversed for the output.

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

mm output for depths: if activated the depths or the thicknesses are output in mm

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

amplitudes: The amplitudes are also reported if the option is activated. You may output the amplitude as the true values (option **unnorm.**) or as normalized values. Normalizing the amplitudes may be done using **low**, **mean** or **high** resolution. Values which do not differ significantly from the mean value (dependent from the selected resolution), are characterized by the symbol o, lower amplitudes by the symbol -, higher amplitudes by the symbol +. The symbol position defines the size of the deviation.

traveltimes: if activated the two-way traveltimes for each layer will be output in addition. Note: no averaging is done for the traveltimes.

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

marker comments: if activated the comment markers are reported at the given positions in addition (see also edit comment markers).

statistic summary: If activated a statistic summary is added at the end of the report or at the end of each block defined by a marker comment (option marker comments activated). The summary contains the following information:

1. line: statistics for comment marker

2. line: length of section

3.- n. line: statistic summary for each layer. Each line contains the information for one layer:

average absolute amplitude

average amplitude

average thickness

standard deviation of the average thickness

max. thickness deviation between two adjacent picks

number of cores

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

If the option summary comment data is activated the program calculates the standard deviation of the thicknesses of the individual layers even if the output is depth.

comments: if activated a freely definable comment block is output in addition at the very beginning of the report. The comments are stored on the file ReportComment.txt under the project directory after having started the report output. In analogy the comments are automatically loaded from this file when activating the option comments.

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

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

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

xyz-coordinates: if activated the xyz-coordinates of the receivers stored in each trace header of the current profile are written out within the 2. , 3. and 4. column (see also trace header Edit menu - chap. 1.9.6). This allows you for example to write out non equidistant xy-coordinates or the xy-coordinates, if the profile is not orientated into one direction (x or y).

layer codes: if activated the layer codes (see global code within the Pick save MenuItem, chap. 1.12.2.1) are reported in addition.

Separator: allows to define the separator type. If comma has been selected the file extension of the report will be csv.

The small buttons "s" and "g" allow to save and load the actual settings on the file LayerShowReportParameters.txt stored under the program directory.

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

batch create: allows to generate reports for a selectable number of layershows, the current parameters are used for all reports. After having activated the option the layershow files are asked for.

With the option **for several lines** activated the reports are generated for several profile lines using the layershow with the same filename like the profile. After having activated the option the profile line files are asked for (only processed files stored under procddata).

close: terminates the window.

1.12.3.3 Create LayerShow velocities

This option allows to calculate laterally variable layer or mean velocities and to store these values together with the current distances on an ASCII-file. You may manually input the layer (mean) velocities or calculate the values based on multioffset data (see also Layershow multioffset picks, chap. 1.12.3.5) or on the reflection amplitudes (see also Layershow amplitude calibration, chap. 1.12.3.4) or read them from an ASCII core file (option coredata/1D-models). The resulting layer (mean) velocities are stored on a file with the extension VLA under the path ASCII under the projectpath. This velocity file may be used within the Create LayerShow (chap. 1.12.3.1 - option option velocity file activated). The file has the following format:

```
comment line
comment line
1.distance v-layer1 v-layer2 ..... v-layer100
2.distance v-layer1 v-layer2 ..... v-layer100
....
n.distance v-layer1 v-layer2 ..... v-layer100
```

If the first 4 characters within the second comment line are "mean" the velocities are used as mean velocities within the Create LayerShow menu.

The given velocities refer to the layer above the layer boundary. You may edit the file and change any value.

After choosing the option a window is opened with the following input options:

start position, end position, increment: specify the start and end coordinate in the set dimension for the velocity file. Increment specifies the desired increment between subsequent distance values for the velocity file. If the velocities are calculated based on the amplitudes a mean value is calculated if more than one pick exists within this distance increment. It is recommended to use a quite large increment in order to balance possible statistical amplitude variations.

Reference choice: allows the input or the calculation of the reference amplitude needed for the calculation of the velocities based on amplitude calibration.

Possibilities:

manual: manual input of the true amplitude, this means the entered amplitude must be corrected for all necessary effects (e.g. gaincurve, geometrical spreading).

reference picks: calculation of the reference amplitude based on a picked onset, e.g. the reflexion from a metal plate. The current activated correction options (see below) are used. You have to consider that the gaincurve of the current line is used for the gain curve correction, this means that the reference line and the current data line must have been acquired with the same gain curve and the same input parameters (sample increment, number of samples). The mean value is calculated from all reference picks.

pick files: allows to load the different pick files for the amplitude calibration or for the multioffset picks of the individual layers (see option layer velocity). If the velocities for all layers are entered manually no pick files must be loaded. The maximum number of selectable files is 200.

Within the "**velocity box / layer**" panel you may specify how the layer velocities for each layer are determined:

layer number: enter the layer number

layer velocity determination: input if the layer velocities above the given

boundary are entered

manually (option **manual**) or

calculated based on the pick amplitudes and a

reference amplitude (option **amplitude calibration**) or

calculated based on

multioffset data (option

multioffset picks) or taken

from core data (option

core data/1D-models).

If **manual** input is activated

only one value for each layer

may be entered for all

distances. For layer

boundary 1 the amplitude

calibration is not available.

If **amplitude calibration** is

activated the velocities for

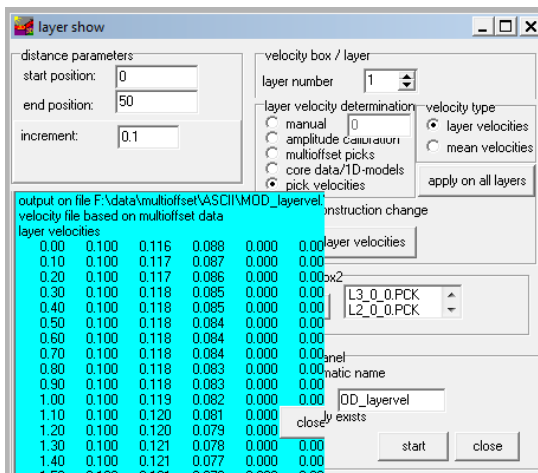
the individual layer are calculated from the selected pick file corresponding to that layer using the correct options (see also Layershow amplitude calibration, chap. 1.12.3.4).

If **multioffset picks** is activated the velocities for the individual layer are calculated from the selected pick files using an automatic adaptation of the picks with different offsets (see also Layershow multioffset picks, chap. 1.12.3.5).

If **core data/1D-models** is activated the velocities for the individual layer are taken from a core-file ((see also View Menuitem, chap. 1.4 - option Core data/1D-models) to be chosen from the filelist. An automatic interpolation between the values at the different core positions is performed. If no information for a distinct layer is given at all the velocity is set to 0. If only no information is given at distinct positions again an interpolation (extrapolation) between the given values is done.

If **pick velocities** is active the velocities stored with the picks are used for the generation of the 2D-velocity field. If layer velocities is activated (see figures on the right) the mean pick velocities will be automatically transformed into layer velocities. Remark: the velocities for the 1. layer are identical for layer and mean velocities. With the option **correct dip** activated the program automatically takes into account the dips of the layer defined by the chosen pick file. The option **slope size** given in traces allows a smoothing for the slope determination. For the dip correction the velocity of the overburden layer must be taken into account. With the option **use max. velocity** the maximum velocity within the whole overburden layer package will be used.

To be considered: The different layer velocity types can be combined - this means for example that you may choose manual for the first layer, multioffset picks for the second one and core data for the third one and manual for all the



others. Restrictions: a combination of multioffset picks and amplitude calibration is not allowed.

velocity type: input if layer velocities or mean velocities are written out to the VLA-file. The 2. comment line of the VLA-file contains the information if the velocities are assumed to be layer (interval) or mean velocities. If amplitude calibration is used always layer velocities are written out. The conversion to layer velocities (option layer velocities activated) based on pick velocities or multioffsets picks is done using the formula for vrms-velocities.

apply on all layers: activate this option in order to apply the current chosen layer velocity determination item on all layer numbers. To be considered: if the velocity item "manual" is chosen you still have to specify the velocity for every layer.

use construction change: if activated a query appears for redefining the manual velocities whenever a construction change marker has been found (see also global settings chap. 1.2.1). If the option coredata/1D-models has been chosen for any layers no interpolation will be done within the area bordered by the construction changes unless two or more coredata are located within this area.

set all layer velocities: the option predefines the manual velocity input to the specified value for all layers for which the manual velocity determination type is chosen.

Within the "**correction for amplitude calibration**" panel you may specify some options which only act for the amplitude calibration:

correct gain curve: With this option activated the gaincurve stored in the fileheader of the current line is used for the amplitude correction. One must consider that before picking the line must not be corrected in time and the starting time (timebegin) must not be changed.

correct transmission loss: With this option activated the transmission losses T , when the wave is entering a layer boundary, are corrected ($T=1-r^2$ - with r -reflexion coefficient).

correct geom.spreading: With this option activated the geometrical spreading is approximately corrected. The amplitudes or the picks are multiplied with the travel time. This is an approximation for a constant velocity. The starting time of the line must be set to 0 and must be equal to that time when the impulse is generated.

Within the "**control panel**" you are starting the calculation:

filename: allows the input of an arbitrary name. The file receives the extension VLA and is stored in the directory ASCII.

start: starts the velocity calculation and the data are stored on the given file name. The velocities are displayed within a list box. Closing this box terminates the window.

close: terminates the windows.

1.12.3.4 LayerShow amplitude calibration

Under very restrictive assumptions (see below) it is possible to calculate approximately the layer velocities from the reflexion amplitudes.

Under the assumption that the reflexion coefficient is only dominated by a velocity contrast the vertical incidence reflexion coefficient is given by:

$$r = (v_2 - v_1) / (v_2 + v_1)$$

with v_1 - velocity of the upper layer
 v_2 - velocity of the lower layer

Knowing the amplitudes of the incoming and reflecting wave and of v_1 the velocity in the lower layer is given by:

$$v_2 = v_1 * (A_0 + A_1) / (A_0 - A_1)$$

with A_0 - amplitude of the incoming wave
 A_1 - reflexion amplitude

Normally it is difficult to determine A_0 . In the case of a radar measurement with the source in the air you may use the reflexion from a metal plate as the reference. This reflection has a phase shift of 180 degrees. Under the assumption that the initial impulse has a negative sign, v_2 is defined as:

$$v_2 = v_1 * (A_0 - A_1) / (A_0 + A_1)$$

with A_0 - amplitude of the incoming wave with 180 degrees phase shift
 A_1 - reflexion amplitude

The measured amplitudes do not correspond to the true amplitude values of the incoming and reflecting wave. They are dominated by several factors:

- geometrical spreading
- transmission losses passing through a layer boundary
- damping

In addition the reflexion is influenced by

- rough reflector
- transmission zone instead of a pure discontinuity
- thin layers

All these effects do have a great influence on the dynamics of the wave field. Most of these effects cannot be corrected for. Approximable the following corrections can be applied:

- geometrical spreading (see option `geometr.spreading`)
- transmission losses when passing a layer boundary (option `transmission loss`)

To be considered: In any case you have to pick the correct phase because the polarity of the amplitude determines whether a velocity decrease or increase is given.

Because of the huge number of possible effects which cannot be corrected for the velocity determination based on the amplitudes must be very carefully interpreted.

Example of processing flow:

- data processing - you must be sure that no filter or editing step is used which influences the succeeding amplitude correction. Therefore no static correction must be used, if you want to correct for the gaincurve and the geometrical spreading. Additionally the timegain must not be changed except you entered the correct values into the gaincurve in the profile header. Additionally the starting time must be equal to that time the impulse is sent (this also holds true for the reference line).
- picking the reflexions - the first boundary must be the reflexion from the air-surface boundary. All layers must be subsequently numbered starting at 1.
- picking the reference reflector, e.g. the metal plate reflection. To be considered: the same gaincurve and the same input parameters (sample increment, sample number) must be used for the reference line as for the data line.
- calculation of the velocities based on the pick amplitudes using the option `create velocities` - activate the wanted corrections.
- Generate the layer-show under the option `create`. Use for velocity choice the option `velocity file` for the time-depth migration with activated or deactivated option `layerinterpolation`.
- Generate a report under the option `create report` using the identical amplitude corrections as for the velocity calculation.

1.12.3.5 LayerShow multioffset picks

If constant offset lines with different offsets have been acquired you have the possibility to automatically determine the layer velocities along the profile from these multioffset data.

First you have to pick the layer boundary for all the different lines within the Pick Menu (chap. 1.12.2.). To be considered: the source receiver offset must be stored within the fileheader (see option S/R-distance in chap. 1.9.3). The picks must be stored separately for each layer and for each offset within the Pick save Menu (chap. 1.12.2.1). The layernumber must be entered and must be the same for each offset.

Then you may calculate the layer velocities within the Create LayerShow velocities menu (chap. 1.12.3.3) using the option multioffset picks. First you have to select the wanted pickfiles.

For each layer you may switch between the manual input or the automatic determination based on the multioffset data but you have to consider the following: if you are using the manual input for the upper layers you should use the mean velocity type because otherwise the program is not able to calculate layer velocities from the multioffset picks (see warning message below). A combination with amplitude calibration is not allowed.

The program automatically determines the best mean velocity fit for the picks with the different offsets (at least two picks with different offsets must exist). The option velocity type determines if layer (interval) velocities or mean velocities are written out on the VLA-file.

The following warning messages may appear:

- for some layerpoints only picks with one offset have been found: this may happen if the layer was not picked at the same positions for all offsets e.g. if the layerboundary is not continuous. Therefore a suggestion in this case: pick the layer boundary for the first offset and save it. Load the picks for the next offset data and only change them and do not include new picks.
- transformation into layer velocities not always successful because no upper picks found: this may happen if the velocity type is set to "layer velocity" and the program is not able to transform the calculated mean velocities to layer velocities because the upper boundary is not specified at distinct points, e.g. if the upper boundary is not continuous. Please use in this case the velocity type "mean velocity".

A **dip correction** may be applied both for multioffset picks and pick velocities.

With the option **correct dip** activated the program automatically takes into account the dips of the layer defined by the chosen pick file. The option **slope size** given in traces allows a smoothing for the slope determination. For the dip correction the velocity of the overburden layer must be taken into account. With the option **use max. velocity** the maximum velocity within the whole overburden layer package will be used.

The following equation is used for a recursive determination of the layer velocities from the NMO-velocities v_{nmo} derived from the multioffset data.

t_0 time of the normal ray

$$v_{NM\phi}^2 = \frac{1}{t_0 \cos^2(\beta_0)} \sum_{i=1}^N v_i^2 \Delta t_i \prod_{k=1}^{i-1} \left(\frac{\cos^2 \alpha_k}{\cos^2 \beta_k} \right)$$

Δt_i time of the normal ray i

α_i angle of incidence i

β_i - angle of refraction i

1.12.4 CMP-processing

With the CMP-processing option activated a complete CMP-processing of single shots stored in one file is possible. The main possibilities:

- setting and controlling of the CMP-processing geometry
- sorting to shot, CMP, geophone or offset with the possibility of a fast change of the sorting parameter
- interactive development of a 2-D velocity model (CMP velocity analysis) and stacking of CMP, shot or geophone sorted data (CMP-processing stack).

The menu CMP processing allows the sorting of data from single shots on a profile to CMP, to common receivers, shots or offsets. The sorting is done with ascending shot coordinates. The program searches the min./max x and y-coordinates of the shots and depending on the largest difference a sorting direction is determined. Then the shot numbers are sorted based on ascending x- or y-shot coordinates depending on the found max. difference. The same is done for the receiver numbers. Therefore it may occur that the display sorting within the CMP-processing both for the shots and the receivers is different from the original one.

The CMP processing is based on a file containing all shots. Please use the option conversion sequence/combine lines/shots within the import menu for the conversion of the single external formatted shot data to the internal REFLEX-format. This option allows an automatic combination of the selected data into one file (see also Import ConversionMode).

The geometrical settings may be edited, e.g. if they are not or not complete read from the original files. For defining the geometry click on the option **geometry** in the tab control. Editing can be done manually or by applying a predefined standard geometry (see also CMP-processing geometry).

The program uses the geometrical settings to create a so-called INDEX-file. This file contains all information on the locations of the CMP's, the shots and the geophones.

If the geometry is defined, you may easily switch the sorting to **CMP**, **shot**, **receiver** or **offset** simply by activating the corresponding option. The ensembles to be processed (e.g. display, velocity analysis, stack) must be predefined using the following parameters:

min.number: enter the number of the first ensemble (CMP or shot or ...) to be shown or stacked

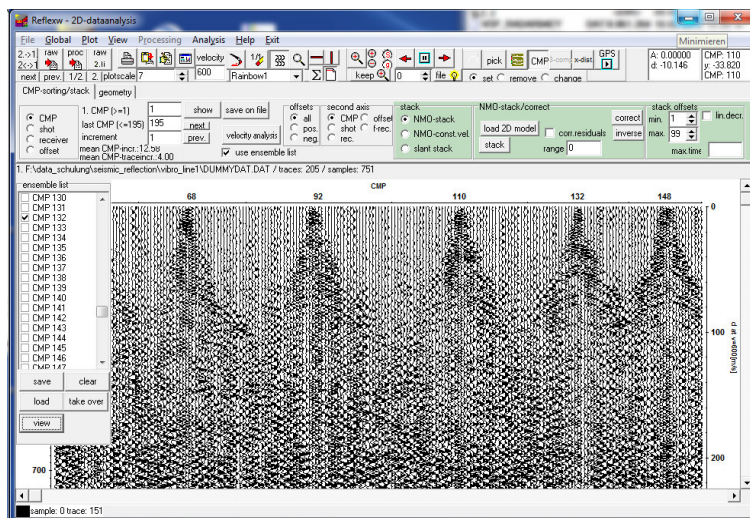
max.number: enter the number of the last wanted ensemble (CMP or shot or ...) to be shown or stacked

increment: enter the increment between the ensembles to be shown or stacked. The increment determines which ensembles between min. and max. number are maintained for further processing.

Activating the option **show** replots the data window based on the current ensemble settings.

The options **next** and **prev.** allow to switch to the next or previous ensemble(s). The start and end ensemble will be increased (decreased) by the entered increment.

It is possible to restrict the display of the ensembles to those with only positive or negative offsets between shot and receiver (switch box **offsets** with items **all**, **pos.** and **neg.**).



With the option **use ensemble list** it is possible to define the wanted ensemble within a list. Click within the list on any ensemble you want to display and which shall be used for the further velocity analysis. It is possible to **save** the actual list and to **load** it. One list will be saved for each project directory. **Take over** allows to take over the actual settings within the min./max.number parameters into the list. **Clear** clears the list. **View** allows to display the ensembles activated within the list. The options show, next and prev. always act on the min./max number parameters. This allows a fast setting of the wanted ensembles within the list.

The option **save on file** stores the chosen ensembles on a freely selectable filename (will be stored under procddata within the extension 00t).

The **second axis** box controls which second sorting variable shall be displayed. If the variable corresponds to that defined within the sorting box no second axis is shown. The second sorting variable is displayed using a 90° rotated font. By default the second sorting variable are displayed for each trace. If the traces are too narrow in order to display each variable the program automatically calculates a factor which controls the number of variables to be displayed. The option f-rec allows to display the original field record number if defined.

velocity analysis: activating this option opens the CMP velocity analysis menu for the selected ensembles. The program automatically creates all necessary files (filenames, e.g. CMP_0020.DAT for CMP 20) and jumps into the menu CMP velocity analysis. The program asks for the trace increment of the individual ensembles. The default increment is calculated from the trace increment stored within the fileheader. You must enter the correct increment because the velocity analysis is based on a constant increment. The program calculates the mean trace increment between all traces within the individual ensembles and shows it on the top of the input query. If there is a jump within one ensemble, e.g. at the shotposition, zero traces are automatically included.

For each ensemble you may use all options of this menu to create 1-D velocity models which may be combined together to a 2-dimension model which can be used for a subsequent stacking.

Stacking (see also CMP-processing stack) may be done either using a NMO-stack or a slant stack.

1.12.4.1 CMP-processing geometry

Within this panel the manual editing of the geometrical settings or the applying of a standard geometry to a selectable number of traces is enabled.

The program creates a so-called INDEX-file on the basis of the geometrical settings, containing all informations on the locations of the CMP's, shots and geophones.

To be considered: the sorting within the geometry table is done with ascending shot coordinates. In the case that the data have been acquired with descending shot coordinates (this means in negative direction - 1. Fieldrecord has the greatest shot coordinate and last fieldrecord has the smallest one) the shotnumbers are inverse to the fieldrecord numbers or tracenumbers. If a standard geometry will be used for such a case, you have to enter the largest shot coordinate for the shot start (according to the shot coordinate of fieldrecord 1) and a negative shot increment. The same holds true for the receiverline if the first geophone has the greatest coordinate and the last geophone the smallest one.

Two different **standard geometries** are implemented: moving line and fixed line.

Activating **moving line** allows you to define the geometry for a geophone line moving with the shots. After the selection of this option you must enter the parameters in the lower left panel. **Nr.of channel** is the number of channels per shot. **First** and **last trace** define the range on the profile for which the standard geometry should be valid. This gives you the opportunity to declare standard geometries for different ranges of the profile. Instead of the first and last trace it is also possible to define the range using the original **field record numbers**. The first and last trace number are automatically changed when entering the field record numbers and vice versa. **Shotstart** is the position of the shot of the first ensemble, **shot increment** is the distance between successive shots and **shot offset** is the distance between a shot and the profile in perpendicular direction. **Receiver increment** is the distance between successive receivers (positive for receivers sorted from left to right and negative if the first receiver has a larger coordinate than the last receiver) and **receiver offset** is the distance between a geophone and the profile in perpendicular direction. **First receiver, left shot receiver, right shot receiver** and **last receiver** define the position of the geophones with respect to the shot.
example:symmetrical geometry with 24 channels and a receiver increment of 1 meter: first receiver: -12, left shot receiver: -1, right shot receiver: 1, last receiver: 12. In contrast a one-sided geometry with a receiver increment of 4 m and 24 channels would be e.g.: first receiver: 4.00, left shot receiver: 0.00, right shot receiver: 4.00, last receiver: 96.00.

Following different spreads are summarized with

x1: first receiver
 x2: left shot receiver
 O: shot (origin)
 x3: right shot receiver
 x4: last receiver

The inputs are given in line direction (see option standard line direction).

Two-sided spread with positive receiver increment:

x1 x2 O x3 **x4**

Two-sided spread with negative receiver increment:

x4 x2 O x3 **x1**

One-sided spread left to the shot with positive receiver increment:

x1 **x2=x4** O x3 set to O

One-sided spread left to the shot with negative receiver increment:

x4 **x2=x1** O x3 set to O

One-sided spread right to the shot with positive receiver increment:

x2 set to O O **x3=x1** **x4**

One-sided spread right to the shot with negative receiver increment:

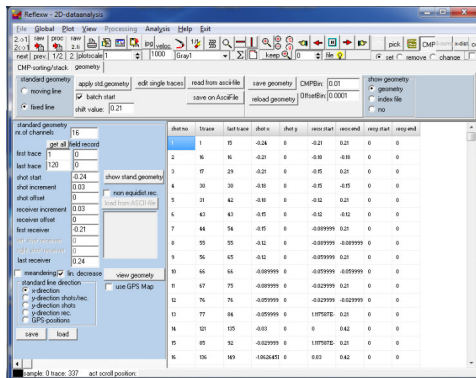
x2 set to O O **x3=x4** **x1**

For the one-sided spreads x3 (spread left to the shot) or x2 (spread right to the shot) must be entered as a dummy value (outside the true receiver line) in order to completely define the geometry. The program automatically redefines this position.

Activating **fixed line** allows you to define the geometry for a fixed geophone line for different shot points. The input parameters are the same as for moving line except the input of the receiver positions. You only have to specify the first and last receiver position. First and last geophone are the absolute coordinates in contrast to moving line where relative coordinates (to the shot) are used.
 example: 48 channels and a receiver increment of 2 meter: first receiver: 100, last receiver: 194.

With activated option **meandering** the start and end coordinates of the receivers are flipped for each 2. shot. Activate this option if the receivers are acquired meandering.

With activated option **lin.decrease** the number of channels are assumed to be not fixed but the number decreases by 1 with increasing shot number. Some devices like the Proceq PD80 are using such a data storing where for the first shot 15 (7) receiver channels right to the shot position will be acquired, for the next shot only the 14 channels right to the shot, for the next only 13 ,...., and for the last shot only 1 channel will be stored.



The radio box **standard line direction** allows to define the direction of the standard geometry.

With the option **x-direction** (default) activated the line is assumed to be orientated in x-direction. All the other options should only be activated if no further CMP-processing steps are done because for some CMP-processing steps the line direction is always assumed in x-direction.

With the option **y-direction shots/rec.** activated the total line (receivers and shots) is assumed to be orientated in y-direction. Use this option for example to define the geometry of a two boreholes transmission measurement.

With the option **y-direction shots** activated the shots are assumed to be orientated in y-direction. Use this option for example to define the geometry of a borehole containing the shots and the receivers placed at the surface.

With the option **y-direction rec.** activated the receivers are assumed to be orientated in y-direction. Use this option for example to define the geometry of a borehole containing the receivers and the shots placed at the surface.

With the option **GPS-position** activated the geometry will be created based on GPS-coordinates. Precondition is that GPS-coordinates are present within the traceheaders for each field record, this means that the shot and receiver traceheader coordinates must contain the coordinates of the GPS-unit. You may enter a moving line relative to these GPS-coordinates. The input of this relative geometry is the same like for x-direction but the option shot-increment is replaced by **offset smooth** and the profile direction is not X but it is automatically determined from the GPS-coordinates whereby the option offset smooth controls this automatic definition of the profile direction from the given GPS-positions. All input parameters are relative to the profile direction, this means that the input of shot start and first, left, right and last receiver correspond to the profile direction and shot offset and receiver offset perpendicular to it. By the parameter **offset smooth** a range is given over which a smoothing will be performed in order to determine the profile direction. The parameter is given in number of shots and not in the distance dimension. The total smooth range therefore calculates from the entered offset smooth multiplied with the average shot increment. Such a smoothing is useful if the GPS coordinates exhibit some small scale variations which do not reflect the true profile direction. A useful value for the smooth range is for example the total receiver length or some part of it.

The following example shows a standard geometry with the GPS-unit located on the ship and the shots 30 m behind this unit with a lateral offset of 6 m. The first receiver is located 15 m and the last receiver 84 m behind the shot (one-sided spread left to the shot with negative receiver increment as the first receiver has a larger coordinate than the last receiver which results in a different internal sorting of the receivers as the original ones).

The shot and receiver traceheader coordinates originally contained the coordinates of the GPS-unit. After applying this standard geometry with 24 channels the shot and receivers positions for each trace have been calculated. If a CMP-stacking shall be done afterwards the CMP-bin must be set to a sufficiently large value (e.g. half the mean shot increment) because normally the individual CMP's do not exactly fit if GPS-coordinates are used. The distance axis of the resulting stack also starts at 0 if GPS coordinates are used and an optional smoothing of the coordinates will be done if the distances significantly deviate from the linear distance.

CMP-sorting/stack geometry

standard geometry
☒ moving line
☐ fixed line

edit single traces read from ascii-file save geometry CMPBin: 0
 apply std.geometry save on AsciiFile reload geometry OffsetBin: 0,0001

show geometry
☒ geometry
☐ index file
☐ no

standard geometry
 nr. of channels 24
 field record

first trace 1 3782
 last trace 12288 4293

shot start -30
 offset smooth 20
 shot offset -6
 receiver increment -3
 receiver offset 0
 first receiver -15
 left shot receiver -15
 right shot receiver -12
 last receiver -84

show stand.geometry
 view geometry

standard line direction
☐ x-direction
☐ y-direction shots/rec.
☐ y-direction shots
☐ y-direction rec.
☒ GPS-positions

save load

shot no	ltrace	last trace	shot x	shot y	recx start	recx end	recy start	recy end
1	1	24	6116253	648709,5	6116253	6116253	648709,5	648709,5
2	25	48	6116256	648712,6	6116256	6116256	648712,6	648712,6
3	49	72	6116260	648715,6	6116260	6116260	648715,6	648715,6
4	73							
5	97	24	6116235	648684,8	6116169	6116220	648632,2	648679
6	121	48	6116238	648687,9	6116172	6116223	648635,3	648682,1
7	145	72	6116242	648690,9	6116176	6116227	648638,3	648685,1
8	169	96	6116245	648693,4	6116179	6116229	648640,9	648687,6
9	193	120	6116249	648696,2	6116183	6116234	648643,6	648690,4
10	217	144	6116252	648698,5	6116186	6116237	648645,9	648692,7
11	241	168	6116255	648701,9	6116189	6116240	648649,3	648696,1
12	265	192	6116258	648704	6116192	6116243	648651,5	648698,3
13	289	216	6116262	648707,4	6116196	6116247	648654,8	648701,6
14	313	240	6116265	648709,9	6116199	6116249	648657,4	648704,2
		264	6116268	648713,3	6116203	6116253	648660,7	648707,5
		288	6116271	648715,8	6116205	6116256	648663,2	648710
		312	6116275	648719,5	6116209	6116260	648666,9	648713,7
		336	6116277	648722	6116212	6116262	648669,5	648716,3

With the option **non equidist.rec.** activated the relative receiver line is assumed to consist of non equidistant receivers - the geometry must be loaded from an external ASCII-file with each line containing the relative receiver position. Of course the ASCII-file may also contain equidistant receiver positions. This freely definable receiver line may be applied both to a moving and to a fixed line.

standard geometry
 nr. of channels 24
 get all field record

first trace 1 0
 last trace 336 0

shot start -4,5
 shot increment 10
 shot offset 0
 receiver increment 5
 receiver offset 0
 first receiver 0
 left shot receiver 0
 right shot receiver 0
 last receiver 115

show stand.geometry
 view geometry

☒ non equidist.rec.

standard line direction
☒ x-direction
☐ y-direction shots/rec.
☐ y-direction shots
☐ y-direction rec.
☐ GPS-positions

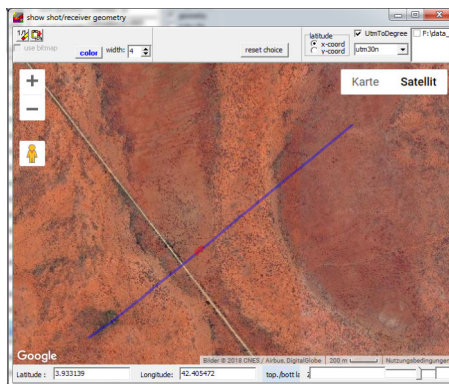
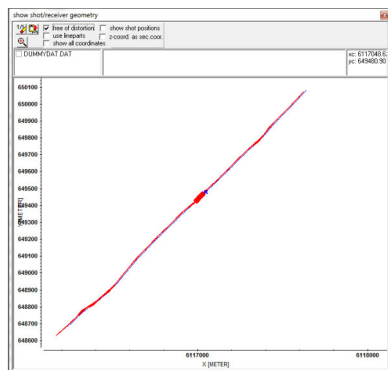
save load

40.00	9
50.00	11
57.00	12
60.00	13

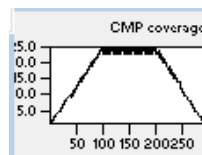
The option **show stand.geometry** allows to sketch the standard geometry.

The option **view geometry** allows to view the geometry of all shots. For this purpose click on any shot number within the table. The actual shot (in blue) and the receivers (in red) are highlighted. You can use the up/down cursors in order to move through all shots.

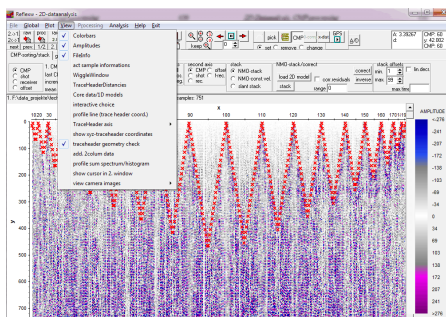
With the option **use GPS Map** activated the geometry will be displayed within a GPS Map. Again the actual shots and receivers may be displayed separately by clicking on any shot number (picture on the right).



The **CMP coverage** will be displayed in a separate window when clicking on view geometry. The display may be enlarged by clicking on the coverage graph.



An alternative possibility to check the geometry is given using the option **Traceheader geometry check** under View. It allows to view the shot-receiver geometry. The actual velocity within the plotoptions is used in order to define the traveltimes between shot and receiver used for the display. The options may be used for a check based on any type of data sorting (example shows a CMP-sorting).



The option **save** allows to store the current standard geometry settings on disk. The filename automatically receives the extension STD and the data are stored under the path ASCII under the projectdirectory. The option **load** allows to load a previously saved standard geometry.

For applying the defined standard geometry to the current profile it is necessary to choose the option **apply std.geometry**. With activated option **batch start** the entered geometry will be applied on all traces using a new geometry for each block defined by first and last trace and updated geometry values based on the entered **shift value**. The option is only useful for a fixed line which will be updated for each fixed line block. This option allows you to update the geometry of different fixed lines within one step.

The option **read from ascii-file** allows you to read the geometry from an ASCII-file. The ASCII file must exist under the path ASCII and must have the extension DST. Each line of the ASCII file contains the following 6 informations:
tracenumber distance Shot-X-Pos Shot-Y-Pos receiver-X-Pos receiver-YPos
 Defining the tracenumber gives you the opportunity to read the geometry only for a distinct part of the data.

The option **save on AsciiFile** allows to write the geometry to an ASCII-file with the extension DST (format see option read from ascii-file above).

The option **edit single traces** allows you to change the geometry of each trace individually. When activating this option the trace header Edit menu will be opened.

The switch box show geometry controls the table output in the lower right box. Activating **geometry** a table containing all current geometrical settings of the individual shots are shown.

Activating **index file** displays the current index number for the shots, receivers and offsets.

The parameter **CMP-bin** defines the bin size for defining matching CMP's. All CMP's falling within this binrange are combined together. Use a value greater than 0 if the geometry of the 2D-line is chosen in such a way that too few CMP's match. The geometry of the individual traces is not changed by this parameter. The parameter only acts on the CMP-sorting of the individual traces. Use the option save geometry in order to apply a new CMP-bin value.
 A very big CMP-bin allows to build so called **super-gathers** which might be useful for an easier velocity adaptation.

The parameter **offset-bin** defines the bin size for defining matching offsets. All offsets falling within this binrange are combined together. The geometry of the individual traces is not changed by this parameter. The parameter only acts on the offset-sorting of the individual traces. Use the option save geometry in order to apply a new CMP-bin value.

The option **save geometry** saves the current settings of the geometry within the fileheader and the traceheaders of the current profile. The old geometry stored within the traceheaders will be lost of course. If you want to preserve this information you should use the option save on ASCIIFile before changing and saving the new geometry.

The option **reload geometry** loads the geometry from the traceheaders of the current profile. Possible changes of the geometry which have not been saved within the traceheaders will be lost.

1.12.4.2 CMP-processing stack

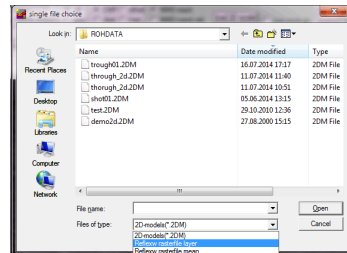
Stacking is possible for data sorted to CMP, shot or geophone depending on the current selection of ensembles. First you have to enter the wanted sorting ensemble (e.g. CMP) and enter the ensemble range for the stacking (e.g.: 1. CMP and last CMP and increment whereby the increment should be set to 1).

Two different stack possibilities are available:

NMO-stack: This option allows the normal moveout stack of sorted data. The dynamical correction of travel times is done according to a preselected 2-D velocity model (option **load 2D model**). The filetype within the filechoice menu allows you to specify whether the 2D-velocity distribution is obtained from the **CMP-analysis** (filetype 2D-models (*.2DM)) or directly read from a Reflexw formatted **rasterfile**. The **2D-model** must have been constructed within the menu CMP velocity analysis. The **rasterfile** may either contain the mean velocities (filetype Reflexw rasterfile mean) or the layer velocities (filetype Reflexw rasterfile layer). The velocities (amplitudes) must be in m/ns or epsilon for GPR data and in m/s for seismic or acoustic data. The rasterfile may either be a depth model (y-axis always meter) or a twoway travelttime model (y-axis corresponds to the two way travelttime). The x-axis is always the distance. After having activated the option **stack** the filename as well as the trace increment of the stacked section is queried (see comment at the end of this chapter). The stack is stored under the path procddata with the extension 00t.

With the option **corr.residuals** activated an automatic correction of the residual statics is applied for the NMO-stack (option stack) and the NMO-correction (option correct). The procedure is as following:

1. A stack trace is created for each ensemble based on the given 2-dimensional velocity distribution.
2. A crosscorrelation is performed for each NMO-corrected trace within each ensemble and the corresponding stack trace.
3. From this crosscorrelation a timeshift value is derived for each trace. The max. allowable shift is defined by the parameter range (enter in the given timedimension).
4. The NMO-corrected traces are shifted by the corresponding timeshift values and the stacking (NMO-correction) is performed.



NMO-const.vel.: This option allows the normal moveout stack of sorted data based on a constant **velocity**. After having activated the option **stack** the filename as well as the trace increment of the stacked section is queried (see comment at the end of this chapter). The stack is stored under the path procddata with the extension 00t.

slant stack: This stack sums all information along straight lines through the origin and assigns this sum to the vertical travel time. This is done for various slopes or velocities within a given range (see below). The summation is done automatically for each velocity and each vertical travel time. No weighting is applied. No velocity analysis prior to stacking is needed. The method works if along straight lines phases to be enhanced are coherent and phases to be decreased go through destructive interference. The following input parameters are needed:

min.vel.: enter the minimum velocity for the slant stack in m/ns or m/s respectively

max.vel.: enter the maximum velocity for the slant stack in m/ns or m/s respectively

dv: enter the velocity increment for the slant stack in m/ns or m/s respectively. The velocity range (max.vel-min.vel) is subdivided into discrete velocities based on this increment.

After having activated the option **start** the filename of the stacked section as well as the trace increment of the stacked section are queried (see comment at the end of this chapter). The stack is stored under the path procddata with the extension 00t.

The image shows a software dialog box titled "stack_offsets". It has four input fields with up/down arrows: "min." (value 1), "max." (value 99), "lin.decr." (value 5, with a checked checkbox to its left), and "max.time" (value 400).

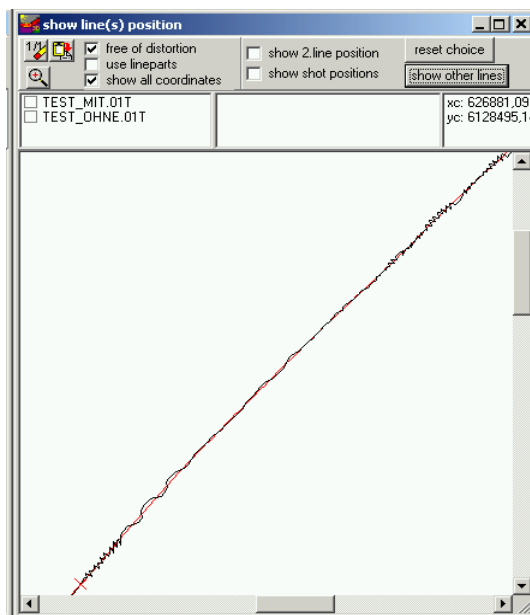
Within the **stack_offsets** panel the offset range may be redefined. By default the min. and maximum existing offset numbers for the current dataset are taken. Here you may restrict the offset range by entering a higher number for **min.** and/or a lower number for **max.**

With the option **lin. decr.** activated a factor greater 1 may be entered which allows to linearly decrease the max. offset from the entered **max. time** to time start. Example: timerange: 500 ms, max. offset set to 50, lin.decr. set to 5 and max.time set to 200. The max. offset for the start time is 10 and increases linearly up to 50 for the time of 200 ms. At time 100 ms the max. offset is 30. For all times greater than max. time (between 200 and 500 within this example) the max.offset of 50 will be used.

To be considered for the stack section:

The default value of the trace increment is given by the number of stack traces and the coordinate range. If this mean traceincrement does not equal any increment between successive stack traces the resulting stack traces are obviously not equidistant. This may happen for example if the shot interval is not equidistant for all shots. In this case the program (from version 3.03) gives a warning message and asks if equidistant traces based on the mean traceincrement shall be made from the non equidistant stack section. If this procedure is cancelled the resulting stack section not equidistant and you must activate the plotoption TraceHeaderDistances (only enabled for the wiggle mode) or you must make the stack section equidistant afterwards.

Especially in the case of GPS-coordinates and CMP stacking the positions of the resulting CMP's may exhibit significant small scale variations. Therefore an optional smoothing of the coordinates will be done if the distances significantly (more than 0.5 %) deviate from the linear distance. The example on the right shows the result of a non smoothing and the smoothing (red line) of a stacked CMP section based on GPS coordinates.



The option **correct** allows a NMO-correction based on the chosen 2-dimensional velocity model (2D-CMP model or Reflexw rasterfile). No stacking is performed, therefore the number of traces remains the same but the sorting of the data may have been changed according to the chosen sorting ensembles (CMP, shot, receiver or offset). After having activated the option **correct** the filename of the NMO-corrected section is queried. The section is stored under the path prodata with the extension 00t.

The option **inverse** allows to perform an inverse NMO correction. The option might be useful if you had performed some filter operations for nmo corrected data and you wish to go back to the non corrected form (e.g. for a stacking,...).

1.12.5 edit comment markers

With the option **edit comment markers** activated you may edit and remove existing comment markers and insert new comment markers. Comment markers are displayed at the top of the profile if the option **show marker** within the **PlotOptions** menu is activated. If the comment marker text is equal to the construction change marker text (see global settings, chap. 1.2.1) a vertical bar is plotted in addition.

The comment markers are stored in the **comment.fil** file, which is located in the project directory. Each line number in the **comment.fil** file corresponds to a 16-bit value stored in the **Reflexw** trace header, and the comment stored in the **comment.fil** file is assigned to that specific trace. With the option **add. filename for comment.fil** within the global settings menu activated the filename **comment.fil** for the comment markers will be replaced by the actual profile filename (without extension) plus **_comment.fil**, e.g. **file0001_comment.fil**. Thereby each profile file will receive an own comment marker file.

After having activated the option a new panel and a table at the lower left corner of the primary profile appears.

With the option **load comment markers** you may load the existing comment markers into the table. The first column shows the trace number, the second the distance and the third the comment. The symbol "x" within the first row indicates the comment marker. It is possible to edit the comment shown in the third column. The comment markers are shown as blue rectangles within the profiles in addition.

With the option **load distance markers** you may load the existing distance markers into the table. Again the first column shows the trace number, the second the distance and the third the comment. No symbol "x" within the first row indicates the distance marker. Editing within the comment cell is enabled. The distance markers are shown as white rectangles within the profiles in addition.

The option **create** allows to create new comment markers. The new comment markers are shown within the table.

With the option **batch** activated the option **create** allows to create and save new comment markers for a selectable number of datafiles. After having activated the option the wanted datafiles have to be chosen (multiple file choice using the **ctrl** or **shift** key). If the comment markers are created based on ASCII files the ASCII-files must have the same filename (except the extension) and the extension **txt** and must exist under the directory **ASCII** under the projectdirectory. Example: **c:\data\test\procddata\file01__00.t -> c:\data\test\ascii\file01__00.txt**
If the option **ASCII XY-coord.** has been chosen (see below) the ASCII-file type is asked for (either **XY-coord.** files or **hypack file**) in addition.

With the option **keep exist.** active the existing comment markers will be kept if new markers will be created. Otherwise already existing comment markers will be removed. Existing distance markers will be kept (option replace all exist. will be automatically deactivated).

Creating the marker positions is based on:

equidistant: With this option activated the option generates equidistant comment markers based on the distance markers and the entered increment. Each distance marker is the new starting point for generating equidistant comment markers in-between two successive distance markers. By default the current construction change marker text (see global settings menu, chap. 1.2.1) is taken for the comment marker text.

ASCII distance: with the option ASCII distance activated the distance positions and the marker texts of the new comment markers are read from an ASCII-file (one line equals one distance value). 3 different formats are supported:

1. If the chosen ASCII-file has the extension TAB another format is used which corresponds to the format for saving the comment marker table on an ASCII-file (see also option save table):

tracenummer distance markerincrement comentstring

The markerincrement will be ignored. If all distances are 0 the tracenummer will be used. Otherwise the tracenummer will be ignored. Use this possibility for example if you have acquired parallel lines with a multichannel system where the antenna have different start positions and therefore the comment markers are only correctly positioned for some files. Save the commentmarkers of the correct file on the tab-file using the option save table and load the other files, reset the existing comment markers if necessary and create the comment markers using this option.

2. If the chosen ASCII-file has the extension MAR the format for saving the marker increments for the processingstep "marker interpol" (see chap. 1.11.6.1) is used:

tracenummer markerincrement comentstring

If all markerincrements are 0 the tracenummer will be used. Otherwise the first marker is positioned on the first trace. The positions of the following markers are calculated from this position and the entered markerincrements.

3. In all other cases the following format is used:

distance commentstring

ASCII XY-coord.: With the option ASCII XY-coord. activated the comment markers are read from an ASCII-file with each line containing the XY-coordinates of the comment marker and the comment itself. The program searches the nearest traceheader coordinates (rec. x-pos. and rec. y-pos.) within the profile for the given XY-marker coordinates and determines the corresponding tracenummer and distance.

Two different formats of the ASCII-file are supported:

1. comment markers XY-coord. files:

X-coord. Y-coord. Commentstring

2. comment markers Hypack files:

latitude longitude Commentstring

The latitude is converted into the Y-coord.(rec. y-pos.) and the longitude is converted into the X-coord. (rec. x-pos.) both using the following formula (e.g. value: 422214.486):

a. 422214.486 -> this is 42 deg and 22.14486 minutes

b. 42 degrees to minutes -> 2520 minutes

Now the minutes of a. and b. are added and multiplied by a factor of 10000:

$(a.+b.)*10000 = (22.14486+2520)*10000=25421448.6$

cur. picks: converts the current picks in comment markers. The individual comment is automatically determined from the picknumber and a given start no.

After having activated the option create the new comment markers are shown with the **table**. Clicking on any trace number cell changes a comment marker to a distance marker and vice versa.

filenames: the option allows to save the filenames (without path) of e.g. camera images within the comment markers. The filenames must have the following form:

The last 8 characters of the filename specify the trace number position within the profile, e.g.: 0100003396.jpg - the string 00003396 is used in order to determine the tracenummer position of the filenames stored as a comment marker. The option might be used for example for viewing camera images (option view/view camera images/marker, chap. 1.4).

A warning message appears if two or more markers have been found for the same trace number. In this case either the option keep exist. must be deactivated or the individual markers at this same positions must be removed.

Start/end: With the option start/end activated distance markers at the start and at the end of the profile will be added.

The option **move** allows to shift the loaded markers by a selectable increment. The option **replace all exist.** must be activated if distance markers have also been chosen. With the option **batch** activated the option move allows to shift the markers for a selectable number of datafiles within one step. After having activated the option the wanted datafiles have to be chosen (multiple file choice using the ctrl or shft key). All existing comment and distance markers will be loaded for each profile and shifted by the given increment (option replace all exist. will be automatically activated).

The option **reset** allows you to reset the table and therefore to remove all comment markers. If a text has been entered below the reset button only those markers with this comment text will be removed.

The option **set CC comments** sets the construction change marker text (see also global settings menu, chap. 1.2.1) to all blank comment markers.

The option **sort table** allows you to sort the table with ascending trace numbers.

The option **save table** allows to output the comment marker table to an ASCII-file (extension TAB under the directory ASCII under the projetdirectory). Such an ASCII-file may also be used for creating new comment markers for another file using the distance informations for the postions of the comment markers (see option create/ASCII-file).

Activating the option **save** saves the current settings of the comment markers. The tracenumbers are used for the positions of the markers. Only the changes concerning the comment markers are saved. Still existing distance markers remain unchanged if the option **replace all exist.** is disabled.

Activating the option **save 2.lines** saves the acutal comment markers onto other profiles to be selected from the file dialog. Only the changes concerning the comment markers are saved. Still existing distance markers remain unchanged if the option **replace all exist.** is disabled. With the option **distance based** activated the comment markers will be saved for the 2. line based on the distances instead of the tracenumbers.

The option **replace all exist.** controls if the distance markers remain unchanged or not. If activated all existing markers will be replaced by the markers within the table including the distance markers.

It is possible to set a new marker and to remove an existing one using the left mouse button within the profile. With activated option set you can insert a new marker at any position clicking the left mouse button within the profile. With activated option remove you can click on any marker shown in the profile in order to remove it from the table. It is not possible to insert or to remove a marker within the table.

1.12.6 adjust trace gain

Activating this option allows you to individually adapt the gain of the single traces. The option is only available for data stored with the new 16 bit integer or new 32 bit floating point format (see also Import format specification, chap. 1.5.4) The option `tracegain` within the `plotoptions` menu (`PlotGain/Filter`, chap. 1.7.5) controlling if the gains are used for the display is automatically activated.

After having chosen the option `adjust tracegain` a table appears showing the traceheader coordinates and the gain of the individual traces. The gains can be adapted either within the table or interactively within the data using the left or right mouse button. The left mouse button decreases the gain of the wanted trace by 10 % whereby the right mouse button increases the gain by 10 %. The table will be automatically updated when pressing one of the two buttons. `tog` shown within a table and can be changed manually. The option `save changes` saves the current settings of the traceheader coordinates. The option `reload from file` reloads the traceheader coordinates and gains from file.

1.12.7 3-component analysis

The option allows the interpretation of 3 component data. Precondition is that the traceheader items component (identical to the item field record no.) and the Ensemble-nr. are set. The x-component (component =1) is assumed to be orientated to the north and the y-component (component =2) to the east. The z-component is stored under component =3. The sorting of the traces must be always x,y and then z for the first ensemble and so on for the following ensembles.

For the special configuration dipole antenna and two frame antenna it is assumed that the dipole antenna has been stored under the x-component and the two frame antenna must be stored under the y- and z-component (see also option dipole/frame).

Construct 3-component data within the import menu:

Set the sequence processing to **multi components**. The final REFLEXW 3-component datafile will be constructed from several original datafiles containing multicomponent data. Each original datafile must contain one single multicomponent dataset. An ASCII-file under the path ASCII under the projectdirectory (filename multicomponent.*) must exist with each line containing the following informations:

filename(without path) declination/north inclination nr.ofcomponents angle1 angle2 angle....

The inclination is the angle to the vertical. One z-component must exist. The angle of this component must be set to -90°. The order of the components within the original file is defined by the angles.

The max. total number of components (incl. the z-component) is restricted to 9. The program searches those 2 components with the angles next to the north (comp1 and angle1 below) and to the east (comp2 and angle2 below) and a rotation of these components into exact north (0°) and exact east (90 °) will be done. The same holds true for the z-component (comp3 below) which will be rotated exactly to z.

```
comp1 =
comp1*cos(angle1+declination)+comp2*sin(angle1+declination)+comp3*sin(inclination)
comp2 =
comp1*cos(angle2+declination)+comp2*sin(angle2+declination)+comp3*sin(inclination)
comp3 = comp3*cos(inclination)+(comp1+comp2)*sin(inclination)
```

Example of a 3-component original dataset consisting of two files, which will be finally combined to one ReflexW 3 component datafile:

xp040000.sg2 20.0 0.0 3 -90.0 0.0 90.0

xp050000.sg2 30.0 0.0 3 -90.0 0.0 90.0

Example of a 7-component original dataset consisting of two files, which will be finally combined to one ReflexW 3 component datafile:

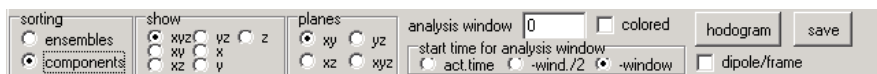
xp060000.sg2 40.0 0.0 7 0.0 30.0 60.0 90.0 120.0 150.0 -90.0

xp070000.sg2 50.0 0.0 7 0.0 30.0 60.0 90.0 120.0 150.0 -90.0

It is also possible to combine files with different number of components into one single file.

Interpretation of a 3-component datafile:

If a 3-component file exist it is possible to interpret it within the **3-component analysis** panel within the 2D-dataanalysis.



The option **sorting** controls the display of the data.

With the option **ensembles** activated the data are plotted into one window with each ensemble after the next. Each ensemble consists of the 3 different components.

With **components** activated the data are plotted into 3 different windows with each window containing all traces of one single component (x, y and z).

The option **show** allows to restrict the number of components to be displayed. With xyz activated all 3 components are displayed, activating xy only displays the two components x and y and so on.

The **particle motion** and the **current polarization** angle can be displayed. In order to do so, one has to choose firstly one of the four possibilities of the option **planes** and secondly the length of the travelttime window (option **analysis window**). The start time of the analysis window can be changed. Activating the option **act. time** means that the start time of the analysis window corresponds to the current cursor position. Activating the option **-wind./2** means a shifting of the start time by half the analysis window length (analysis window is centered around the current time) and **-window** means a shift by the analysis window length resulting in an endtime of the analysis which corresponds to the current time. In many cases the option -window will give you a sharper and more precise view of polarization changes.

The option **hodogram** allows you to continuously display the particle motion within the chosen plane when moving the mouse cursor in the data. The linearity factor (1 - completely linear, 0 - circular) and the dominant angle are also determined and displayed when you have chosen one of the 2-dimensional planes. The option analysis window determines the length of the travelttime window for the display and the polarization analysis. If the plane xyz has been chosen, a 3D-cube display of the particle motion is shown.

The option **colored** allows to color the wiggles based on the current polarization angle. Again the option analysis window determines the length of the travelttime window for the polarization analysis. The option is disabled when you have chosen the plane xyz. With the option **use colors f.3.comp.** activated the wiggles for that component which lies outside the polarization plane are also colored using the calculated polarization angles.

The option **save** saves the angles and the linearities on two Reflexw files. The files have the same name like the original datafile plus '_ang' for the angle file and '_lin' for the linearity file and are stored under the path rohdata. Example: Original data: file0002.dat - anglefile: file0002_ang.dat

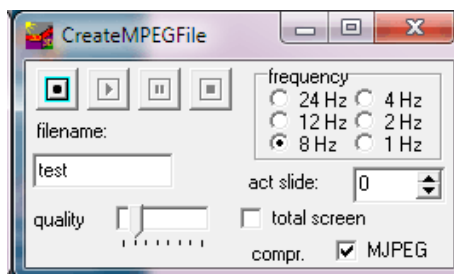
With the suboption **dipole/frame** activated the angles are determined from the dipole antenna (x-comp.) and two frame antenna (y- and z-comp.). This is a very special configuration and therefore shall only be used for this configuration. The

angle of the 3 antenna towards north is stored within the traceheader under CMP(x-pos.). This angle will be subtracted from the calculated one (see also processing option rotate 2 components (chap. 1.11.5.7)).

If picks have been loaded within the 3-component analysis the actual **angles** can be exported together with the picks when using the ASCII-columns format (option angles activated - see chap. 1.12.2.1).

1.12.8 create mpeg file

After having activated the option a new window named create MPEG-file opens which stays on the top. First you must enter the **filename** (without extension). The MPEG-file will be stored under the current projectdirectory with the extension mpg. The option total screen allows to capture the total screen. Otherwise only the primary and (if activated) secondary screen will be captured.



The **compression** is controlled by

the trackbar and the checkbox **MJPEG**. The trackbar allows you to either increase the **quality** (trackbar to the left) or to increase the **compression** (to the right). With the highest compression the filesize will be decreased by a factor of about 4 compared to the highest quality. With the option MJPEG deactivated an additional encoding is used which again reduces the filesize by about 50 %. In this case the computertime for constructing the MPEG-file will be significantly increased and the quality as well will be decreased.

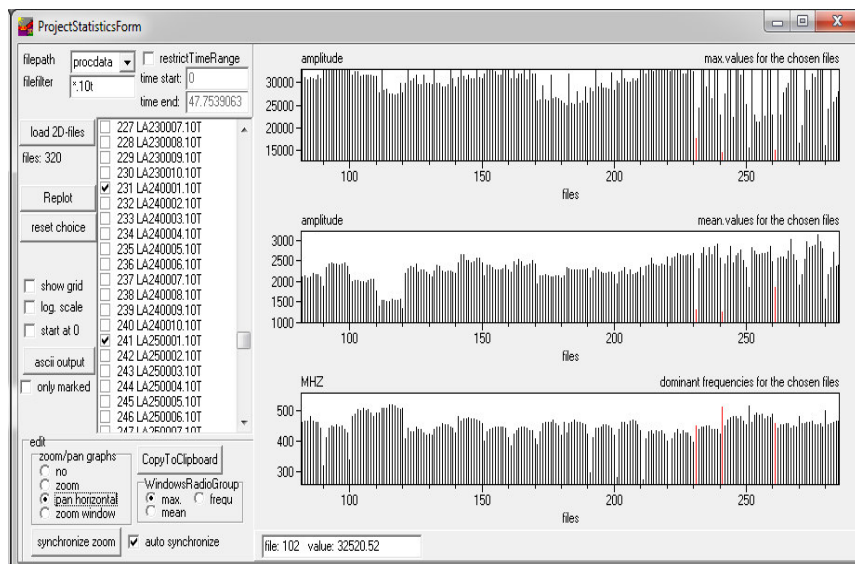
In the frequency Radio Group box you may set the **frequency**, with which the certain pics of the mpeg-file will be shown (8 Hz means 8 pics per sec, e.g.). Start the recording by pressing the **record** button. Now the other 3 buttons (play, pause and stop) are enabled. The **play** button will be automatically activated and the first slide is added to the MPEG movie. As long as the play button is activated each new slide will be added if the actually shown data is changed (e.g. when using the scroll bars or when using the plot option). The current slide number is shown. If you want to interrupt the MPEG-construction use the button **pause**.

The button **stop** finishes the construction and the MPEG-file will be closed.

To be considered: the bitmap size must not be changed. Therefore an error message appears and the MPEG construction will be stopped if the bitmap size, e.g. after having resized the 2D-dataanalysis window, has been changed. The bitmap size is determined when pressing the record button.

1.12.9 2D-lines statistics

Activating **2D-lines statistics** opens a new window where a set of 2D-lines may be analysed in different ways. The window is also available from the 3D-datainterpretation module within the Generate 3D-file from 2D-lines menu. One useful application of the option might be a check of the energy homogeneity between the different 2D-lines, e.g. if an antenna array has been used for acquiring parallel 2D-lines. It also may serve as an overall overview about the amplitude distribution, e.g. before generating a 3D-datafile for which the datavalues should not exceed a value of 32000 (16 bit integer limitation).



The option allows to analyse the maximum and the mean of individual 2D-lines as well as the dominant frequency. The values will be displayed in different windows and may be marked either with the list box or by using the right mouse key in any of the value windows. The marked lines will be displayed in red in all windows and are checked within the listbox. The box symbol allows to specify the type of display - either a **mark** in form of a rhombus or a vertical **line**. The option **reset choice** resets all markings.

Zooming and panning is available. With **zoom** activated enables both to increase and to decrease the magnification factor of the graph. With the left mouse button pressed and held down moving to the left and/or bottom decreases the magnifications and moving to the right and/or top increases the magnification. With **pan horizontal** activated the data display can be panned by pressing and holding the left mouse button. The data image will then follow the mouse movement. The panning is only done in x-direction (file number). With **zoom window** activated a zoom range may be defined. For this purpose press and hold down the left mouse button at one of the corners of the intended region. Thereafter a rectangle follows the mouse cursor indicating the region to be

zoomed. Releasing the left button selects this region and the zoom region has been defined. A zooming in y-direction (amplitude) will be automatically reset when marking any line. Activating **no** resets all zooming and panning.

The analysis of the files may be exported to an ASCII file using the option **ascii output**. With the option **only marked** activated only the analysis of the marked files will be saved.

It is possible to restrict the time range for the analysis using the option **restrict TimeRange** and by entering the wanted time start and end values.

By default the y-axis starts at the minimum value. Activating **start at 0** sets the starting point for all windows to 0. The option **show grid** allows to display a grid. If **log. scale** is clicked a logarithmic scale will be used for the y-axis.

The option **CopyToClipboard** enables you to copy any of the windows to clipboard. The WindowsRadioGroup controls which window will be copied.

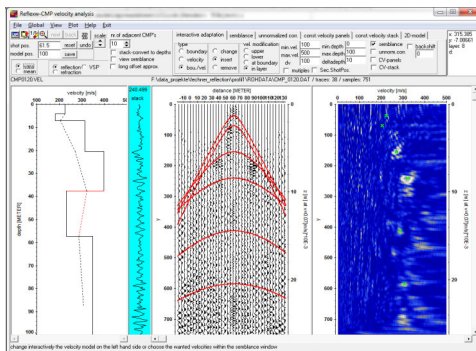
The option **synchronize zoom** enables to synchronize the actual zoom of one window for the other windows. The WindowsRadioGroup controls which window will be used for the synchronization. If the option **auto synchronize** is active the synchronization is automaticall done for zooming and panning (not for zoom window).

2. CMP(1D) velocity analysis

The module CMP-velocity-analysis allows the calculation of a one-dimensional velocity-depth-distribution from CMP- or moveout-data or wide-angle refraction data as well as VSP (VerticalSeismicProfiling) based on different analysis techniques. The adaptation can be done using the original seismic or GPR data or using the picked traveltimes. The original seismic or GPR data are either plotted based on on the overall fileheader coordinates or using the traceheader distances. The traveltimes are always displayed along the distance axis using the distance coordinates stored with the traveltimes.

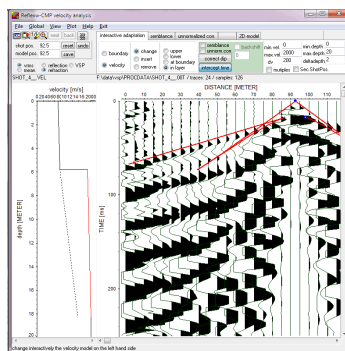
Reflection data analysis:

This is the standard application which is used e.g. for multi offset seismic or GPR data. Also single moveout data, e.g. with a fixed source and a moving receiver line may be analysed in order to extract a one dimension velocity distribution. It is recommended that the datatype of the seismic (GPR) file is of single shot in order to take into account the correct shot and receiver locations.



Refraction data analysis:

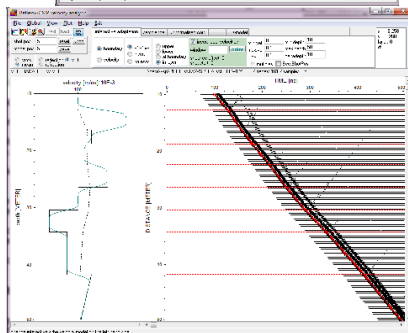
The module also allows a direct interpretation of the first arrival of a refraction data set (single shot with a fixed receiver line). It is recommended that the datatype of the seismic (GPR) file is of single shot in order to take into account the correct shot and receiver locations.



VSP data analysis:

The third data which can be analysed using this module are Vertical Seismic Profiling data. Analogous to the refraction data interpretation the first arrivals can be directly adapted. It is recommend that the datatype of the seismic (GPR) file is of single shot/VSP in order to take into account the correct shot and receiver locations.

The module offers the following possibilities:



- interactive** CMP velocity adaptation of a velocity-model for a CMP- or a moveout-section with continuous indication of the current reflections and/or refractions.
- loading of a second CMP-section** for a parallel adaptation of the reflections and/or refractions
- loading of a zero-offset section** with true distance information for a calibration of the corresponding reflections and/or refractions
- different normal moveout analysis method for a given velocity-interval for reflection data like **semblance analysis**, **unnormalized correlation**, **constant velocity panel** and **constant velocity stack**, interactive choice of a vrms-depth-distribution from the analysis methods, direct reconstruction of the interval velocities, possibility to change the resulting distribution by the model generation module described above
- **intercepttime** analysis for creating a starting model from the intercepttimes of the refraction onsets.
- **corridor stack** for Vertical Seismic Data.
- generation of a CMP 2D-model** based on the resulting 1D-velocity-depth distributions. This 2D-model represents the base for the stacking, the migration and the depth-conversion.

The following options are valid for all analysis techniques:

reflection: if activated the primary and multiple (with option multiple activated) reflections are calculated and displayed. Use this option for the adaptation of reflection data.

refraction: if activated the complete traveltimes including forward and backward travelling waves (reflections and refractions) are calculated and displayed. Use this option for the adaptation of wide angle refraction and reflection data.

VSP: if activated the first arrival traveltimes for a VSP geometry are calculated. Use this option for the adaptation of Vertical Seismic Profiling data. The shot/rec offset and the shot start value are listed within the VSP-box. Both values are taken into account for the velocity analysis.

shotpos. For reflection and refraction data - allows the specification of the current shot location. By default the shot location is taken from the fileheader of the profile. For a correct velocity analysis this value has to be correctly specified.

Shot offset: for VSP-data - allows the specification of the lateral offset of the shot. The difference between shot offset and rec. offset will be taken into account for the calculation of the VSP-traveltime curves.

model pos. For reflection and refraction data - allows the modification of the position of the model within the profile line. The model position is stored within the velocity file. If no velocity file is found, the model pos. is automatically set to the shotpos. The model position is important, if a 2D-velocity model shall be created (see option CMP 2D-model). Please notice that modifications of the model position only are saved after saving the model.

Rec. offset: for VSP-data - allows the specification of the lateral offset of the receivers. The difference between shot offset and rec. offset will be taken into account for the calculation of the VSP-traveltime curves.

save (save model): allows to save the current velocity-depth distribution in a file with an arbitrary name with the extension VEL in the directory ROHDATA in the project directory. By default the file name of the data section without extension is adopted as name.

save model as time,vrms...: allows to save the actual 1D-model using different formats: depth-vrms (depth over root mean square velocity), depth-mean (depth over vertical mean velocity), time-vrms (two-way traveltime over root mean square velocity), time-mean (two-way traveltime over vertical mean velocity), time-interval (two-way traveltime over interval velocities). The model will be saved under an arbitrary filename under the path ASCII using a file extension corresponding to the chosen format.

reset: resets the current 1D-velocity model to default values. All changes not saved will be lost.

Undo: allows to undo the last change of the model.

load data: opens the wanted data section. You may load the data from the rawdata directory (the directory containing the files which are not preprocessed) or from the procd data directory. If you are choosing procd data filter, a filter for the file extension must be entered in order to make the choice easier. The program controls whether a velocity file with the same name like the data file exists. If no velocity file is already loaded, this file will be automatically opened. In the other case a message appears if the new velocity file if existing shall be loaded.

load traveltimes: allows to load traveltime (pick) data without loading any wave data. The program controls whether a velocity file with the same name like the data file exists. If no velocity file is already loaded, this file will be automatically opened. In the other case a message appears if the new velocity file if existing shall be loaded.

It is possible to load REFLEX formatted PCK files as well as
2D-columns ASCII with each line containing
distance traveltime

ASCII-3D-tomography with each line containing (see also chap. 1.12.2.1)

traveltime code x-transmitter y-transm. z-transm. x-receiver y-receiver z-receiver

ASCII-2D-tomography with each line containing (see also chap. 1.12.2.1)

traveltime code x-transmitter y-transmitter x-receiver y-receiver

load data/traveltimes: opens the wanted wavedata section together with traveltime (pick) data (formats see above). You may load the data from the rawdata directory (the directory containing the files which are not preprocessed) or from the procd data directory. If you are choosing procd data filter, a filter for the file extension must be entered in order to make the choice easier. The program controls whether a velocity file with the same name like the data file exists. If no velocity file is already loaded, this file will be automatically opened. In the other case a message appears if the new velocity file if existing shall be loaded.

Load Second Line: The option allows the additional selection of a secondary profile (e.g. another shot or CMP-section or the zero-offset profile). This section is shown below the section to be investigated in a separate window. The travel time curves on the basis of the current velocity model are plotted in both sections.

There are two different possibilities of comparing profiles:

a. comparison with a second shot or CMP-section, e.g. a cross profile, in order to consider anisotropy effects. The option **Sec.ShotPos.** should be activated for this kind of comparison.

b. comparison with the zero-offset profile. This comparison is suitable, e.g. for better recognizing reverberations or large layer dips in the region of the CMP, possibly having an influence on the velocity determination within the CMP or the shot. The option **Sec.ShotPos.** should be deactivated for this kind of comparison, in order to facilitate a correct, distance true comparison of the beginning point of the CMP or the shot with the zero-offset profile.

In both cases each profile is scaled individually in x- and y-direction. Use the zoom option to optimize the distance range of the second line in that way that an optimal comparison of zero-offset profile and shot(CMP)-section is guaranteed.

next: the next ensemble (CMP, shot or receiver group) is chosen. The option is only enabled if the CMP velocity menu is opened from the CMP-processing menu.

back: the previous ensemble (CMP, shot or receiver group) is chosen. The option is only enabled if the CMP velocity menu is opened from the CMP-processing menu.

load model: load any existing 1D-velocity model.

Print: Use this option for printing. Depending on the current settings only the the velocity model together with the data (only primary data) is printed or the semblance or the 2D-model is printed in addition. After having activated the option the print menu opens. Here you may enter the size of the x- and y-axis of the data display.



enter PlotOptions



copy to clipboard



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



enable manual zoom - With the option ZOOM an arbitrary area of the data set can be selected and plotted in full screen size. If a secondary file has been loaded this option can also be applied to the second data set

The area to be enlarged, a rectangle, has to lie within a data set. Pressing the left mouse button you determine a corner of this rectangle and by moving the mouse with pressed button the desired area is opened.

Zooming is only available within the data display and not within the 1D-velocity plot, the semblance and the 2D-model display.

Scrolling is possible in x- and time-direction. Again scrolling is only available for the data display (only primary data). One peculiarity: the timeaxis of the

semblance is directly connected to the data time axis. Therefore scrolling in time direction is also possible for the semblance.



allows to easily switch between pointmode and wigglemode

scale: allows to decrease or increase the multiplication factor for the color-amplitude assignment or for the wiggle size.

EditFileHeader: opens the file header menu for making changes of the file header for the current data file (not for the secondary file).

multiples: if activated the first multiple reflections from the surface are calculated

Sec.ShotPos.: if activated the shot position for the second line is read from the corresponding file header of the second line. This option is only relevant, if a second profile is loaded using the option **load Second line**. Activating this option is reasonable, if two shot(CMP)-sections shall be compared with each other, since in this case the travel time curves always begin at the shot location of the corresponding CMP.

vrms: the root mean square velocity is used for calculating the reflection hyperbola and for determining the layer (interval) velocities from the semblance analysis.

mean: the mean velocity (mean for normal incidence) is used for calculating the reflection hyperbola and for determining the layer (interval) velocities from the semblance analysis.

view semblance: If activated the actual semblance or unnormalized correlation analysis (automatically converted into depth using the 1D-velocity model) which must have been performed first will be plotted into the model (see chap. 2.2).

stack convert to depths: allows to automatically convert the stack trace (option view StackTrace activated) as well as the normal moveout data (option view/nmo corrected ensemble activated) and the stack of the adjacent CMP's into depths. This may allow a more precise interactive definition of the layer depths.

long offset approx.: for the reflection type - if activated a fourth-order moveout equation (Al-Chalabi, Bilas) is used instead of the normal rms equation. The option is useful if a strong vertical heterogeneity of the velocities is given and for long offsets. The option will also be used for the semblance and unnormalized correlation. In this case the vertical heterogeneity is also derived from the actual 1D-model. If no 1D-model is present it might be useful to generate a 1D-model from the semblance analysis (for which the vertical heterogeneity was zero) and then run the semblance analysis again.

$$t(x) = \sqrt{(t_0^2 + x^2 / V_{ave}^2) / (1 + \frac{gx^2}{t_0^2 V_{ave}^2 (g+1)})}$$

$$g = (V_{rms} / V_{ave})^2 - 1$$

nr. of adjacent CMP's.: number of adjacent ensembles for the Const.velocity-stack (see chap. 2.2) or the CMP-stack (option adjacent CMP's stack traces).

The option **View** within the main menu allows to control the view of different characteristics and analysis tools:

Amplitudes: if activated the amplitudes within the data and the analysis windows will be displayed within the coordinates panel.

show second model: If activated you may choose a second model which will be displayed in addition in green.

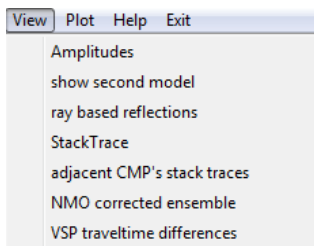
ray based reflections: If checked the reflections based on refracted rays are shown in addition to the mean velocity-based reflections. If the difference between these two reflections is too big, the assumption of the mean velocity is not valid anymore.

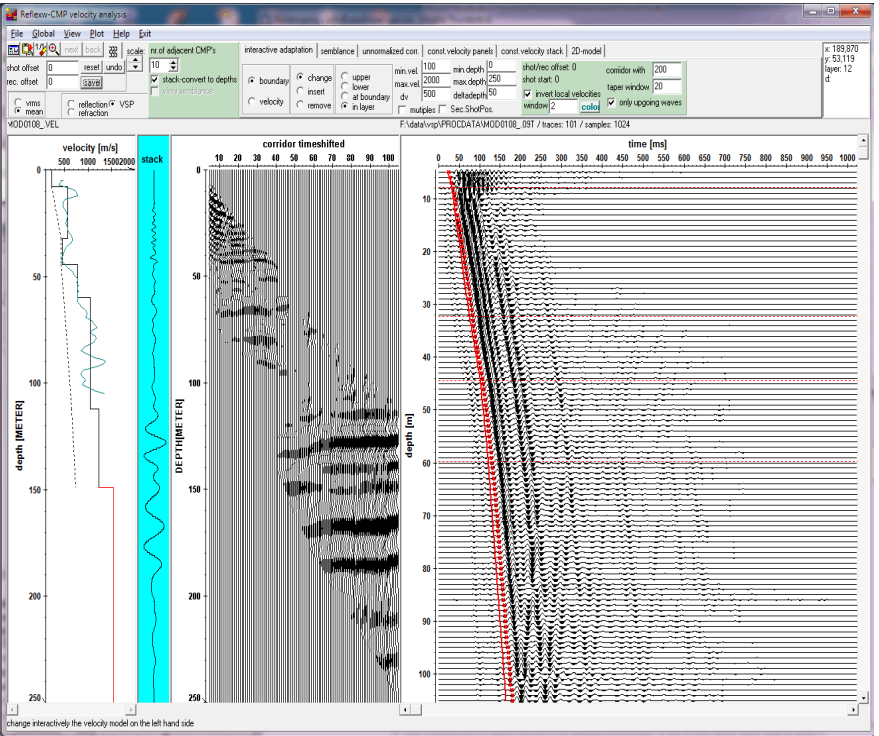
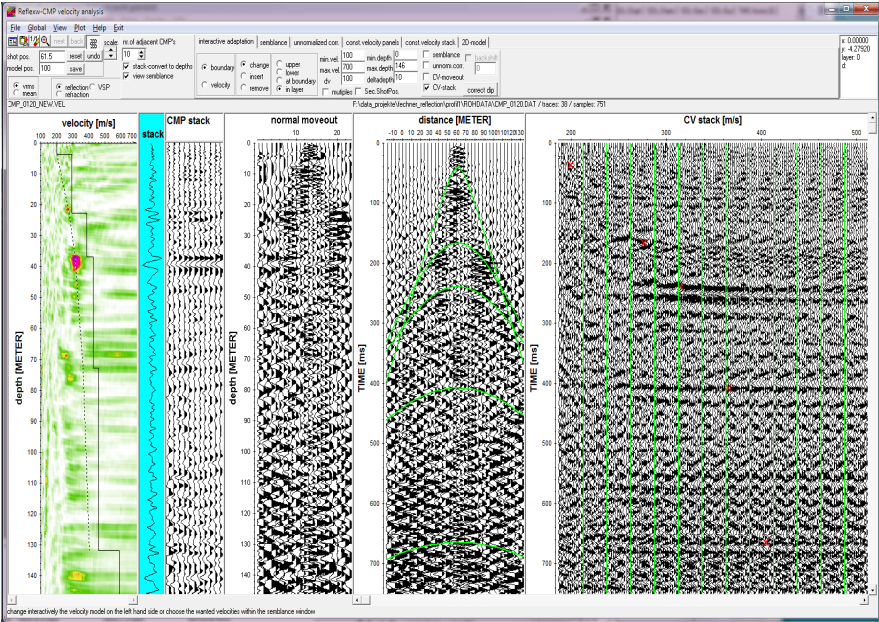
StackTrace: If activated a stack trace on the basis of the current velocity model is plotted in a new aqua window between the model image and the data image. This option may serve as a control possibility for the performed velocity analysis. A complete stacking of various CMPs to a zero-offset profile only can be executed within the module cmp-processing. The stack trace of a loaded second model (see above) will be displayed as well.

adjacent CMP's stack traces: Activating this option allows to display a CMP stack for the number of ensembles given by **nr. of adjacent CMP's**. The program automatically searches the files with identical extension and which are in the alphabetical neighbourhood (example loaded datafile is CMP00120.DAT and nr. of adjacent CMP's is 10 - the program searches for the previous 5 files in the left alphabetical neighbourhood, for example CMP00115.DAT to CMP00190.DAT and for the next 5 files in the right alphabetical neighbourhood, for example CMP00121.DAT to CMP00125.DAT). The files must have identical sample number, otherwise they will be ignored. The actual 1D-velocity distribution is used for the stack. The size of the window changes with the number of used ensembles.

NMO corrected ensemble: If activated the NMO corrected (reflection active) or time shifted (VSP active) section based on the actual 1D-velocity model is plotted in a new window next to the stack trace. For reflection data the number of traces used for the NMO correction depends on the settings within the analysis (semblance,...) panel. According to the option stack trace this option only serves as control possibility for the performed velocity analysis. The NMO corrected section as well as the stack trace will be continuously updated when changing the 1D-model.

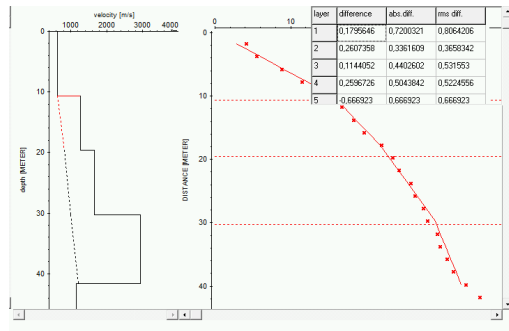
The horizontal size of the window is controlled by NMO scale defined within the global settings menu (chap. 1.2.1). The NMO scale is given in percent of the data windows size. The same plotoptions as for the data window will be used.



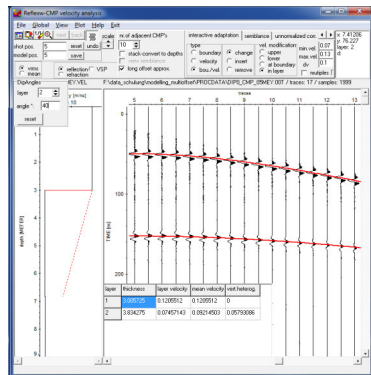


VSP traveltimes differences: option is available for the analysis technique VSP. If activated the differences between the data traveltimes and the calculated traveltimes are determined and shown within a table (positioned on the top right corner, the table can be moved using the right mouse button). The option is only available if traveltimes have been loaded either together with the original wave data or without.

The the total timedifference, total absolute timedifference and the RMS deviation are displayed for each layer. The total absolute timedifference defines the sum of the absolute timedifferences independently from the sign (pos. or neg. difference). This value gives you an estimate of the overall adaptation. The total timedifference defines the sum of the timedifferences taking into account the sign of the differences. This value gives you an estimate whether the mean model leads to too small or too big traveltimes. The RMS deviation defines the Root Mean Square deviation.



model spreadsheet: if activated the actual model parameters (thickness layer velocity, mean velocity and vert.heterog.) will be displayed within a spreadsheet. The vertical heterogeneity (only displayed when the long offset approx. option is activated) may be used e.g. for an improved velocity determination from multi offsets picks within the pick menu.



The option **repaint images** within the global setting menu controls the display refresh. If activated the images will always be repainted when a model change has been done. Deactivate this option if a canvas error occurs. Activate the option if the images need a refresh.

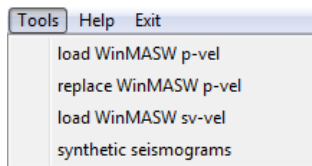
The option **Tools** within the main menu allows:

load WinMASW p-vel: loads a p-velocity distribution derived by the program WinMASW™.

Replace WinMASW p-vel: allows to replace an existing WinMASW™ velocity file by the actual p-velocity model. The option might be useful if a surface wave analysis has been performed using the WinMASW™ program and the p-velocity distribution has been modified within Reflexw by e.g. analysing the first p-wave onsets.

load WinMASW sv-vel: loads a sv-velocity distribution derived by the program WinMASW™.

Synthetic seismograms: allows to generate synthetic seismograms based on the Reflectivity Method - see chap. 2.5.



2.1 CMP velocity adaptation

With the option interactive adaptation activated there is the possibility of an interactive selection and modification of a one-dimensional velocity distribution with immediate visualization of the reflections or refractions or direct traveltimes (VSP) calculated on the basis of the velocity model. The shot location can be freely chosen. Moreover it is possible to optionally compute surface multiples, additionally, and display them, too.

In the upper right corner the coordinates of the current cursor position are shown. There are 3 different analysis windows:

In the **left window** (model window) the current one-dimensional velocity distribution together with the VRMS-distribution (dashed) is shown. Within this window you may modify the velocity model using one of the possibilities described below. The horizontal size of the window is controlled by model scale defined within the global settings menu (chap. 1.2.1). The model scale is given in percent of the data window size.

In the **middle (right) window** (data window) the currently loaded section with the current plot parameters and the reflections or direct waves, calculated on the basis of the current velocity distribution, are plotted. The color and the size of the calculated reflections are taken from the symbol font. If only traveltimes have been loaded for the adaptation the sorting of the traveltimes is normally based on the tracenummer of the original lines. An alternative possibility is given using the option **sort traveltimes by distance**. Activate this option if the calculated curves are not displayed for VSP.

In the **right window** (analysis window only with semblance, unnormalized crosscorrelation, Constant Velocity panel or Constant Velocity stack activated) the semblance or unnormalized correlation or the Constant Velocity panel/stack is shown. The current VRMS-velocities associated with a boundary are marked. Use the symbol font for changing the color and the size of the marks. You may mark any new point and therefore insert a new boundary. This setting is independent from the options insert, change or remove. The horizontal size of the window is controlled by NMO scale parameter defined within the global settings menu (chap. 1.2.1). The NMO scale is given in percent of the data windows size.

The number of discrete points of the calculated traveltimes curves are also defined within the within global settings menu.

boundary: For active option boundary a new layer boundary can be inserted into the one-dimensional velocity model (option **insert** active), an old boundary can be shifted continuously (**change** active) or deleted (**remove** active).

For **changing** the depth of a boundary therefore the options boundary and change have to be activated. Now bring the mouse cursor in the direct proximity of the desired layer and then you can change its depth by keeping the left mouse button pressed and simultaneously moving the mouse upwards or downwards.

For **inserting** a new layer the options boundary and insert have to be active. You set the mouse cursor to the desired location of the model and there insert a new layer boundary simply by clicking the left mouse button. The same procedure allows with active option remove the removal of an existing layer boundary.

velocity: Allows the modification of the velocity where the option change and one of the options upper, lower, at boundary or in layer has to be active. The modification is done by setting the mouse cursor in direct proximity to a layer

boundary, keeping the left mouse button pressed and simultaneously moving the mouse horizontally. With active option **upper** the velocity can only be changed above the selected layer boundary, the corresponding applies for the option **lower**. For active option **in layer** the velocity of the whole layer is changed. For active option **at boundary** the velocity above and below the layer boundary is kept constant and an appropriate variation is permitted.

Due to these manifold modification possibilities discontinuities or arbitrary gradients can be realized.

bou./vel.: If activated both the boundary location and the velocity may be changed using the pressed left mouse button. The velocity will be changed for the whole layer independently from the choice of the vel.modification.

change: Changing an existing layer or velocity (see boundary, velocity).

insert: Inserting a new layer boundary (see boundary).

remove: Removing a layer boundary (see boundary).

upper: For changing the velocities the modification refers to the velocities above the selected layer boundary, i.e. the layer boundary the mouse cursor is next to.

lower: For changing the velocities the modification refers to the velocities below the selected layer boundary.

at boundary.: For changing the velocities the modification refers to the constant velocity above and below the selected layer boundary.

in layer: For changing the velocities the modification refers to the velocity (s) within one layer. These are changed as a whole.

min.vel., max. vel. and **dv** specify the velocity axis range of the 1D-velocity/depth window.

min.depth, max.depth and **deltadepth** specify the depth axis range of the 1D-velocity/depth window.

Reflection data:

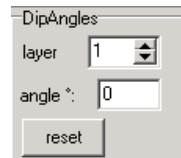
If the options **semblance**, **unnorm.corr.**, **CV-panels** and **CV-stack** (see also chap. 2.2) are active the analysis window is open and changes within the 1D-velocity model are also shown within this window. Additionally you may mark any new point in the semblance analysis window and therefore insert a new boundary.

The option **backshift** is only active if **semblance** or **unnorm.corr.** are activated. It allows to take into account a backshift timevalue which compensates the effect when using delayed wavelet maxima for the semblance or unnormalized crosscorrelation analysis. The timeshift must be chosen in such a way that the resulting reflection curve stems from the very first arrival (see also chap. 2.4).

Refraction data:

The option **intercept time** allows the interactive use of the intercept time method. The option enables to get a first 1D-model very quickly. After having activated the option you must enter the bending points by the left mouse button. The first point is automatically set to distance and time zero. The velocities derived from these settings must increase with depth. After having finished each setting of one straight line the 1D-model is automatically created and shown in the left model window. Deactivate the option when you have finished the intercept time adaptation. It is also possible to store or interactively change this model. The option is the same like the intercepttime adaptation within the velocity adaptation menu of the 2D-dataanalysis window.

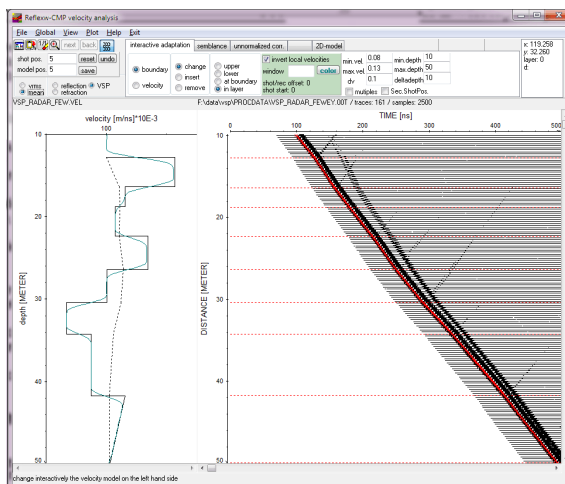
correct dip: if active a new window opens which allows to enter a dip angle in degrees for each layer which will be taken into account when calculating the reflection traveltimes.



VSP data

The shot/rec and shot start values are listed within the VSP-box.

invert local velocities: the option is only available if picked first arrivals have been loaded. With the option activated the local velocities are calculated from the loaded first arrival traveltimes. The velocities are smoothed over a given depth window. The color of the local velocities can be freely chosen.



corridor stack:

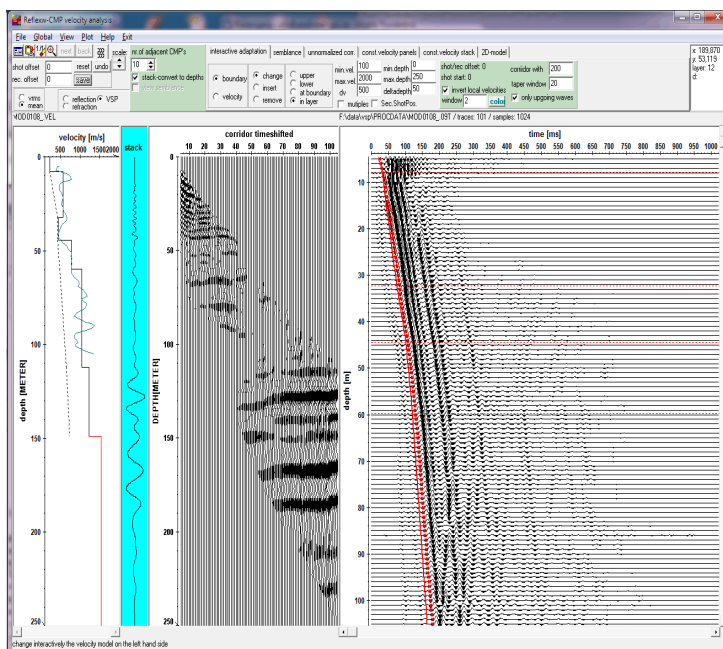
A dynamic correction including stacking of the reflections of a vertical seismic profile (VSP) is supported. The result is a time-shifted two-way arrival times (at the surface) section. The timeshift is done based on the first arrival timeshift over the **corridor width** given in the timedimension. The first arrivals used are calculated from the receiver positions and the actual 1D-velocity distribution. The data are muted above the first arrivals and below the time beginning at the first arrival time plus the corridor width. A taper may be applied around the first arrivals (option **taper window**).

With the option **only upgoing waves** activated a fk-filtering will be done before the timeshift correction/stacking resulting in a section which only contains the upgoing wavefield.

The timeshifted data may be stacked and viewed as a stack trace (option view StackTrace activated) or only viewed (option view/nmo corrected ensemble activated). This corridor timeshifted section will be displayed with the timeaxis as the vertical axis.

The resulted stack trace as well as the timeshifted data may also be timedepth-converted using the option **stack convert to depths**. With this option activated a direct comparison of the 1D-model and the stacked /timeshifted traces are possible.

Remark: the number of samples does not change. If the original data acquisition time is small and deeper reflections are present you may use the processing option time cut within the 2D-dataanalysis before in order to extend the timerange of the original profile.



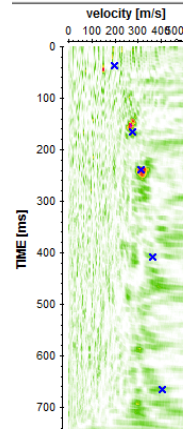
2.2 CMP semblance/Const.velocity analysis

The semblance or the constant velocity analysis allows you to specify vrms velocities within the velocity/time spectrum.

Semblance/unnormalized crosscorrelation:

For a given velocity range and velocity increment (**min.vel.**, **max.vel.**, **vel.interval**) either the **semblance**, i.e. the normalized output-to-input energy ratio, or the **unnormalized crosscorrelation** is calculated within a given **time window** for all times. For the semblance the timewindow is centered around the current time, for the crosscorrelation the timewindow starts at the current time. The summation is done over the given trace range (**1.** and **last trace** and **n.trace**). The calculated quantities are shown after having activated the start button in the right analysis window with the velocity as the x-axis and the travel time as the y-axis. The color assignment can be changed using the PlotOptions menu. You may change the current colorpalette for the semblance display using the option **act. palette** independently from the colorpalette for the current profile.

The velocity range has to be chosen large enough, so that all occurring velocities are comprised. The increment and the number of traces have to be optimized in that way that a good viewing and selection is possible.



$$CC = \frac{1}{2} \sum_t \left\{ \left[\sum_{i=1}^M f_{i,t(i)} \right]^2 - \sum_{i=1}^M f_{i,t(i)}^2 \right\}$$

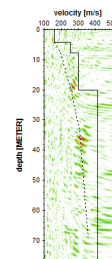
$$= \frac{1}{2} \sum_t \left[s_t^2 - \sum_i f_{i,t(i)}^2 \right],$$

$$NE = \frac{1}{M} \frac{\sum_t s_t^2}{\sum_t \sum_{i=1}^M f_{i,t(i)}^2}.$$

The unnormalized cross correlation CC (see figure on the left, from Yilmaz, 1987) takes into account the true amplitudes and it is therefore the better choice if you only wish to adapt high amplitude reflexions. The semblance analysis NE (see Figure on the left) uses a normalization. That is why the semblance should be used if the analysis is mainly focussed on the correlation of the individual phases and also small amplitude reflexions shall be adapted.

view semblance: If activated the actual

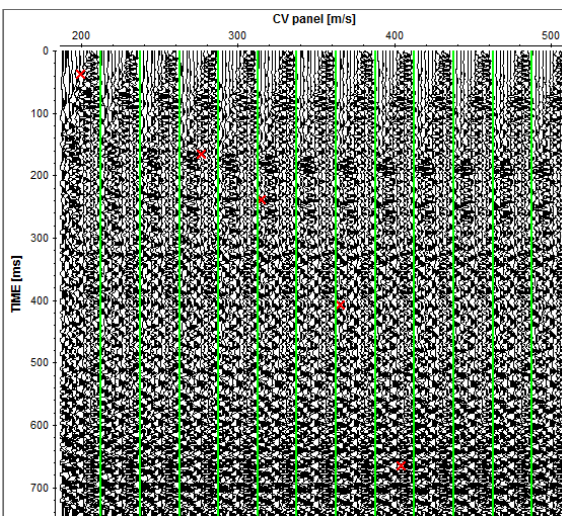
semblance or unnormalized correlation analysis (automatically converted into depth using the actual 1D-velocity model) will be plotted into the model.



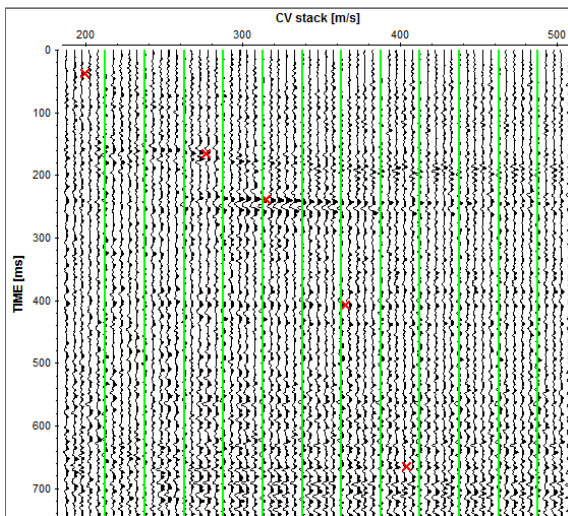
Constant velocity analysis:

With the option **const. velocity panel** activated the NMO-correction is applied for the actual shot or CMP using different constant velocities defined by a velocity range and velocity increment (**min.vel.**, **max.vel.**, **vel.interval**). The moveout correction is done over the given trace range (**1.** and **last trace** and **n.trace**). The results of the different velocities are displayed within the analysis window with each velocity separated by a vertical line and the velocity that results in a flattening of the hyperbolas corresponding to the velocity for a certain reflector.

Analogous to the semblance analysis it is possible to interactively choose the wanted velocities.



With the option **const. velocity stack** activated the NMO-correction is carried out for several adjacent CMP gathers and the NMO-corrected data are stacked. The number of the adjacent CMP's is given by the option **nr. of adjacent CMP's**. The program automatically searches the files with identical extension and which are in the alphabetical neighbourhood (example loaded dtaafilename is CMP00120.DAT and nr. of adjacent CMP's is 10 - the program searches for the previous 5 files in the left alphabetical neighbourhood, for example CMP00115.DAT to CMP00190.DAT and for the next 5 files in the left alphabetical neighbourhood, for example CMP00121.DAT to CMP00125.DAT). The files must have identical sample number, otherwise they will be ignored. Stacking velocities are picked directly from the constant velocity stack panel by choosing the velocity that yields the best stack response at a selected event.



In contract to the semblance/unnorm.correlation analysis the same plotoptions as for the data window will be used for the constant velocity analysis. It is recommended to increase the horizontal size of the analysis window using the global settings option **analysis scale** (in percent of the datawindow size).

After having activated the **start** button, the semblance or the unnormalized crosscorrelation or the constant velocity analysis is performed. After having finished the calculation, the program automatically returns to the CMP velocity adaptation with activated semblance, unnorm.correlation, CV-panels or CV-stack option. It is possible to modify the current 1D-velocity model either within the semblance analysis window or within the velocity model window. The change, insert and remove options work for both the analysis window and the model window. A continuous change with pressed mouse button is also supported within the analysis window. If you want to create a completely new model you must activate the button reset and then insert new boundaries either within the semblance analysis or the velocity model window.

rasterfilename: enter the filename of the Reflexw rasterfile which contains the actual velocity analysis. The file will be stored under rohdata under the current project directory. If blank a dummyfile will be used.

Direct zooming within the semblance analysis is not possible. The zooming in time direction is connected to the zoom within the profile display. A zooming in velocity direction is not possible.

2.3 CMP 2D-model

Within this menu there exist two different methods for creating a 2D-velocity distribution. The first one creates a vrms model, the second one interval layer velocities.

The 2D-dimsional velocity models are created on the basis of 1D-velocity-depth-profiles. This 2-dimensional velocity distribution can be used, e.g. for the migration and depth conversion within the module 2D data analysis or for stacking within the module CMP-processing.

For the creation of the distribution previously created 1D-velocity models are comprised to a 2D-model. For this the position within the profile line for the corresponding 1D-models has to be determined previously (see option **model pos.**). Between these 1D-models a linear interpolation is performed. The 1D-model with the largest model position is extrapolated to greater position values. The 1D-model with the smallest model position is extrapolated to smaller position values of the 2D-model.

2.3.1 2D vrms velocities

The option **create** generates a vrms velocity distribution from the 1D-models. Possible vertical gradients within the 1D-models are ignored. The velocity within each layer must be constant. Otherwise a warning message appears. If vertical gradient are present within the 1D-files is is recommended to generate an interval velocity rasterfile (see chap. 2.3.2).

Within the 2D-model file only the filenames of the chosen 1D-velocity models are stored. That is why changes in any of these 1D-models at a later stage have an influence if the 2D-model file is used for any processing step.

create: allows to create a new 2D-model file under the directory rohdata using the extension 2DM. After activating create you have to select the 1D-cuts, that are going to be used for the 2D-distribution and writes the filenames of these 1D-cuts in the 2D-model file.

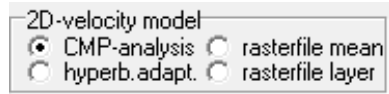
load: loads an existing 2D-model file.

show: shows the VRMS-distribution of the current 2D-model file using a color-coding.

min.dist., **max. dist.**, **min.depth** and **max.depth** define the distance/depth range for showing the 2D-model and for the Reflexw rasterfile if activated..

create corefile: if activated an ASCII coredatfile is automatically created when creating a 2D-model (same filename with extension cor - see also option Core data/1D models under View MenuItem).

If used for a subsequent migration or time/depth conversion the option CMP-analysis must be activated



rasterfilename: enter the filename of the Reflexw rasterfile (vrms-velocities) which is automatically created on the right hand side when starting the creation of the 2D-model. The file will be stored under rohdata under the current project directory. If blank a dummyfile will be used.

Raster-xpoints and **raster y-points** define the size of the Reflexw rasterfile.

2.3.2 2D interval layer velocities

The option **create interval vel.rasterfile** generates an interval layer velocity distribution from the 1D-models. Vertical gradients within the 1D-models are allowed.

After having activated the option **create interval vel.rasterfile** the wanted 1D-velocity models will be queried. A 2D Reflexw rasterfile will be generated from these 1D-velocity models.

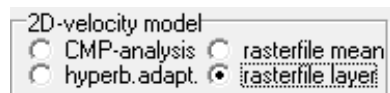
rasterfilename: enter the filename of the Reflexw interval velocity rasterfile. The file will be stored under rohdata under the current project directory. If blank a dummyfile will be used.

min.dist., **max. dist.**, **min.depth** and **max.depth** define the distance/depth range for the Reflexw rasterfile.

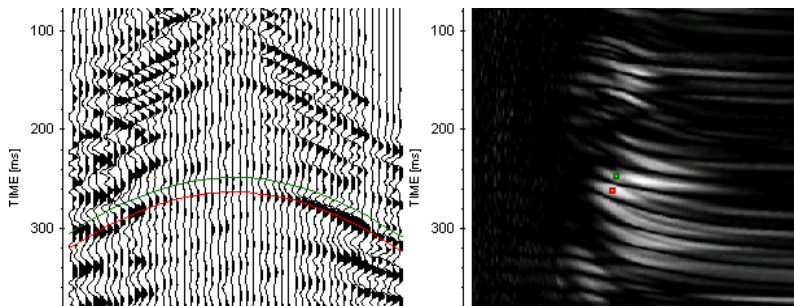
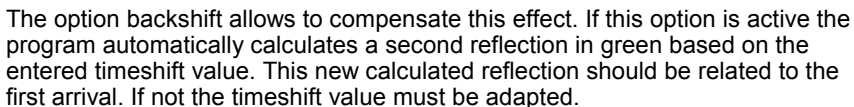
Raster-xpoints and **raster y-points** define the size of the Reflexw rasterfile.

create corefile: if activated an ASCII coredatfile is automatically created when creating a 2D-model (same filename with extension cor - see also option Core data/1D models under View MenuItem).

If used for a subsequent migration or time/depth conversion the option rasterfile layer must be activated.



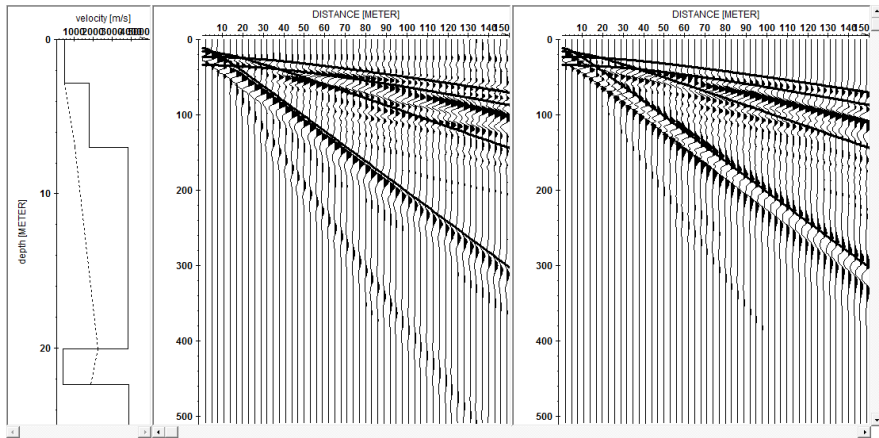
Due to the normally mixed-phase waveform the semblance analysis or the interactive velocity analysis may lead to improper too low velocities. Only the first-breaks yield correct vrms and layer-velocities. If a later arrival (e.g. the second or third half-cycle) is used for the analysis too low vrms velocities result. This is shown within the lower pictures where for two reflectors both the first and a later arrival have been used for the interpretation. The resulting velocities are about 20 % too low when using the later arrivals for the analysis. Especially the semblance analysis may lead to these wrong velocities because the maximum semblance response normally relates to later arrivals (see also Booth et al., 2010).



2.5 calculation of synthetic seismograms

The option **synthetic seismograms** under **Tools** allows to calculate synthetic seismograms based on the actual 1D-model using the Reflectivity Method (Sandmeier and Wenzel, 1986).

The method allows the full waveform simulation for a laterally homogeneous medium. It calculates seismograms for all three components (below only the simulation of the vertical and radial components are displayed).



The actual loaded velocity model forms the p-velocity base for the calculation. The s-velocities are automatically determined from these p-velocities using a Poisson ratio of 0.25. The P-quality factor Q_p is determined from the p-velocity/40. The S-quality factor Q_s is assumed to be $0.75 \cdot Q_p$.

Following you find a description of the input parameters:

calculate: allows to calculate the full wavefield including conversion from P to S waves and vice versa. There is the possibility to restrict the calculation to only P reflections (PP), SV reflections (SS) or P and SV reflections but no conversions (only PP/SS).

Transm.conversion: allows to ignore P to S and S to P conversion during the transmission.

Type: allows to choose either displacement, velocity or acceleration for the type of receivers.

Ver./rad. Comp.: allows to define which component shall be calculated.

Trans.(sh) compl.: defines if the sh-component shall be calculated.

Source: different source types may be chosen.

Explosive should be used if explosives had been used for the wave excitation.

Double couple should be used for an earthquake simulation.

Point source is the standard source for a small scale source radiating P-, SV- and SH-energy degree independent..

Line source simulates a source which only excites in x-z direction.

signal allows to choose any of the following different signal types: Kuepper, Delta, Heaviside, Momentfunction, spike, digitized signal and Ricker.

The digitized signal is taken from the first trace of a Reflexw formatted datafile. A possibly existing starttime within the Reflexw file will not be considered.

Time parameters: DeltaT defines the timeincrement. Tsignal gives the timerange of the signal and together with the parameter extrema nr. the dominant frequency is defined. The dominant frequency is defined by this timerange and the number of extrema (e.g. extrema nr. set to 2 and Tsignal set to 10 ms gives a dominant

frequency of 100 Hz.). The total number of samples is entered by sample nr. and with the timeincrement the total time of given ($\Delta T \cdot \text{sample nr.}$). The sample nr. must be a power of 2. The program automacially increments the number to the next power of 2 value.

The box **frequencies** defines the frequency range for the calculation. Four parameters must be entered starting with min.frequ. and left frequ. (Start of the plateau, between min. frequ. and left frequ. a cosine window is used). The third parameter right frequ. determines the end of the plateau and the fourth the **max. frequency (max. frequ.)**. Between these two points the filter is represented by a cosine-window, too.

The box **phase velocities** defines the phase velocity range for the calculation. Four parameters must be entered starting with min.veloc.. and left veloc. (Start of the plateau, between min. veloc.. and left veloc.. a cosine window is used). The third parameter right veloc.. determines the end of the plateau and the fourth the **maximum velocity (max. veloc.)**. Between these two points the filter is represented by a cosine-window, too. The parameter number defines the number of caluclated rayparameters and thereby the step rate between the velocities. Within **depths** the source and the receiver depths are given in m.

The **receiver line** box defines the first and last receiver position (first dist., last dist.) as well as the traceincrement (increment), the azimuth in degree and the number of receivers (number).

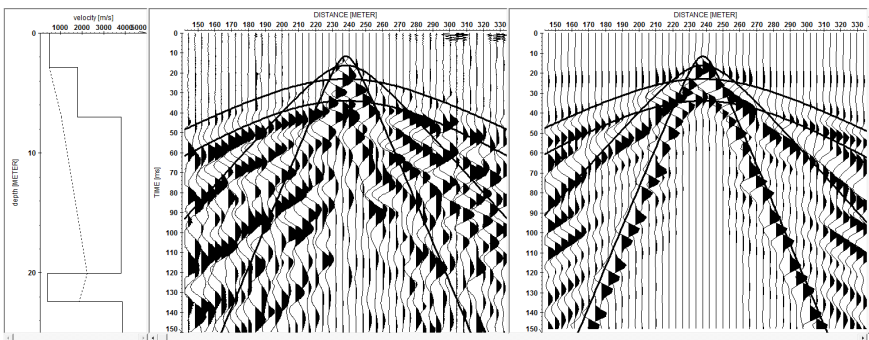
The option **view** allows to specify which component shall be displayed. You may choose between vertical (ver.), radial (rad.) or transverse (transv.) component or both the vertical and radial (ver/rad) components.

With the option **keep loaded data** activated the actual loaded data will be kept and the synthetic data will be displayed within the secondary window.

With the option take over geometry activated the geometry of the primary profile will be taken over for a direct comparison of the primary (real) data and the synthetic data (see picture below).

The Reflexw output fileame must be entered within the parameter **filename**. The filename must have a lenght of at least 8 characters. The vertcial components gets an extension **_ver**, the radial **_hor** and the transverse component **_transv**.

The calculation is started by **start**.



3. 3D-datainterpretation

The module 3D data-interpretation allows the interpretation of 3-dimensional data by displaying x-, y- or z-slices or the full 3D-data volume.

Besides the interpretation of the full 3D-data volume two additional modules are available:

The **scan 3D tool** (see chap. 3.10) allows a fast access to rectangular 3-dimensional GPR- or seismic data which have been acquired along 2D-parallel lines in one or two perpendicular directions without going through the whole process of import, processing and generation.

The **3D-fixed array tool** (see chap. 3.9) allows a fast interpretation of multichannel data with fixed lateral offset between the channels which allows to easily interactively reach each point within the dataset using the mouse either within the individual profiles or timeslices.

A **3D-data file** is a single REFLEX formatted file which consists of several equidistant 2D-lines sequentially stored. All traces belonging to one 2D-line have the same ensemble-number which is stored within the REFLEXW traceheader (see chap. 1.9.6). The 3D-data may be easily constructed from equidistant 2D-lines either during the import (see also Import ConversionMode - option combine lines/shots) or in a later stage within the 3D-datainterpretation menu. The 3D data cube is controlled by only two parameters: the trace increment within each 2D-line and the linedistance between each line.

The data are completely loaded into the RAM of the computer whereby a fast visualization of the data is possible. The max. number of points in each direction (x, y and z or t) is 2048. If the points in one or more direction exceed this boundary Reflexw offers two possibilities rescale and subdivide which can be selected in a separate window which only opens if the 2048 boundary will be reached.

The option **rescale** automatically calculates rescaling factors in all directions where the number of points exceed the 2048 boundary. Example: 4800 points in line direction and 2400 points in time direction. The data are automatically rescaled by a factor of 3 and 2 resulting in 1600 points in line direction and 1200 points in time direction.

The option **subdivide** will only be enabled if only the number of points in line direction exceeds the 2048 boundary. Activating this option generates a number of subdatasets of the complete 3D-dataset in this direction. Two new options next and prev. control the access to the different subdatasets. An example shall illustrate the option:

3D-dataset with 6000 points in line direction, 100 points perpendicular to the line direction and 256 points in time direction. The option subdivision generates 3 different subdatasets each with 2000 points in line direction. The first subdataset is displayed after having activated the option. The option next loads the next subdataset from trace nr. 2001 to 4000 in line direction.

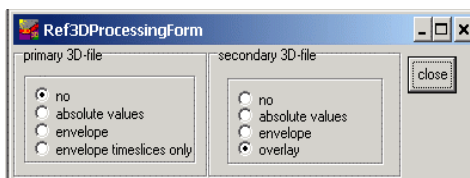
Three different display options are available (option windows, option scroll and option 3D-cube - see below).

It is possible to load a **second 3D-file** in addition. The second 3D-file must have the same number of samples. Otherwise an error message appears. If the number of points in x- or y-directions differ or if the data are sorted in a different

manner (profile directions are different) a warning message appears and it is possible to perform an automatic adjustment of the chosen two 3D-files if the corresponding input query will be confirmed. Then the option **adjust 2 3D-files** (description under analyse, chap. 3.1.5) will be automatically performed and two new 3D-files with the same number of x-cuts as well as of y-cuts will be created and loaded. If only the sorting is different it is also possible to first use the option **FlipXYsorting3DFile** to be found under analyse and then load the second 3D-file. The screen is either horizontally or vertically splitted if a second 3D-file is loaded according to the settings within the plotoptions menu (chap. 1.7.6). If the processing option overlay has been activated within the 3D-processing menu the two 3D-files will be overlaid. The parameter second 3D-file under view controls if the secondary 3D-file is shown or not.

The module also allows you to load several (up to 25) **different 2D-files** in order to display them within the scroll or windows mode (load the 2D-files using the option **Open several 2D-files under 3D-File MenuItem**). The scale of the different 2D-files can vary.

In addition you may construct **single timeslices** (C-scans) from different 2D-lines originating all from one acquisition plane - option **Generate single timeslices under 3D-File MenuItem**.



Using the option **windows** the cuts are displayed in manually scalable windows. The step and smoothing rate may be freely chosen - see also **Windowing 3D-file**, chap. 3.3.

Using the option **scroll** you may continuously move through the 3D-cube either in x-, y- or z-direction using the track bar. Again the step and smoothing rate is freely selectable - see also **Scroll 3D-file**, chap. 3.2.

Using the option **3D-cube** allows the three-dimensional plotting of the 3D-data volume within predefinable spatial limits and an arbitrarily definable observation point. Thus you have the possibility to look at the cube from different angles, e.g. from the front side, the back side or from the side - see also **3D-cube display**, chap. 3.4.

Processing: here you may enter the type of display for both the primary and secondary 3D-file.

no means that the data are displayed as they are.

absoluteValues: if activated the absolute values are calculated and shown. Activating this option is useful for slices for example.

envelope: if activated the envelope of each trace of the 3D-file is calculated and shown. Activating this option is useful for slices for example.

envelope timeslices only: with this option activated the envelope of each trace of the 3D-file is calculated and shown for the timeslices only (option slice activated). For the other cuts (x-cuts and y-cuts) the "normal" traces will be shown. The option is not available for a secondary 3D-file and will also be automatically deactivated for the primary 3D-file if a secondary 3D-file has been loaded.

overlay: The option allows to overlay the secondary 3D-file. The chosen processing option for the primary 3D-file controls how the overlaying is done. With the option no or abs. amplitude activated the two 3D-files are simply added for all 3 cuts. With the option envelope and envelope timeslices only a special algorithm is used for overlaying all cuts (option envelope) or the timeslices only (option envelope timeslices only). The option might be useful for a combined interpretation of two 3D-files with different antenna polarization (GPR-data) or with different orientation of the original 2D-lines.

Use the lower panel for a speed access to some further different options:



enter the project directory



caption 3D-raw: open REFLEX 3D raw data (see also 3D-File Menultem)
 caption 3D-poc/procf: open REFLEX 3D processed data (procf with filter)
 caption 2D-raw: open several REFLEX 2D raw data
 caption 2D-proc/procf: open several REFLEX 2D processed data (procf with filter)
 caption 2D-int: enter the interactive choice menu for opening several REFLEX 2D files

The caption is automatically updated when using one of the following options within the 3D-File Menultem: open 3D-file, open several 2D-files



copy current data bitmap to the clipboard or into a file (suboption file). Different file-formats like BMP, JPEG or TIFF are supported. A clipboard scale factor is queried. A factor larger than 1 allows you to increase the size of the bitmap to be transferred and therefore to increase the resolution.



enter FileHeader Edit



enter PlotOptions



replot current line with current zoom parameters



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



enable magnifying glass function



enable manual zoom - With the option ZOOM an arbitrary area of the data set can be selected and plotted in full screen size - see also Windowing 3D-file or Scroll 3D-file.

The filter Traceinterpol-3DFile allows a resampling of the data in the direction of each line if the number of traces differs in each 2D-line.

Plotscale: The minimum and maximum amplitude values are controlled by the multiplication factor *Amplitudescale*. With a value of 1 for *Amplitudescale* the amplitudes range from -2048 to 2048 for unnormalized data and from -1 to 1 for tracenormalized data. With a Value of 0.0625 the amplitudes range from -32768 to 32768 (see also *Amplitudescale* under *Pointmodeattributes*);).

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

Shift palette for slices: if activated the amplitude range for the color assignment of the point mode will be shifted to only positive values starting from 0 for slices and for the 3D-cube display. Thereby the full actual color palette is available for slices if these have been created using the envelope. The option is also active for the 3D-cube display. In this case the shifted color palette will be used for all cuts.

To be considered concerning the plot options: The individual *Reflexw* modules partly use the same plot options. When using for example the 2D- and the 3D-module simultaneously in order to switch between the 2D and 3D views when making interpretations it might be useful to have different options for these views. Therefore the following settings are separated for the 2D and the 3D-interpretation: manual scaling, color palette, depth axis, show axis.

3.1 3D-datainterpretation MenuItems

3.1.1 3D-File MenuItem

From this menu you may open an existing 3D-datafile, create a 3D-file and edit the file header.

ProjectDir: Selection of a new project. After selecting this option the directories included in the current project directory are listed. From these you can choose the project you want to work with.

Open 3D-file: Use this option to load an existing REFLEX-3D file. A 3D-file consists of several equidistant parallel 2D-lines stored successively within one file. You may load the data from the rawdata directory (the directory containing the files which are not preprocessed) or from the procd data directory. If you are choosing procd data filter a filter for the file extension must be entered in order to make the choice easier. After having chosen the wanted 3D-datafile the 3D-processing menu opens and you have to choose how you want to display the data (see also option processing within chap. 3). The data are completely loaded into the RAM of the computer whereby a fast visualization of the data is possible. If the 2048 points boundary will be reached a windows opens for rescaling or subdividing the too big dataset (see also chap. 3).

Open second 3D-file: Use this option to load a **second 3D-file** in addition. After having chosen the wanted 3D-datafile the 3D-processing menu opens and you have to choose how you want to display the data (see also option processing within chap. 3).

The second 3D-file must have the same number of samples. Otherwise an error message appears. If the number of points in x- or y-directions differ or if the data are sorted in a different manner (profile directions are different) a warning message appears and it is possible to perform an automatic adjustment of the chosen two 3D-files if the corresponding input query will be confirmed. Then the option **adjust 2 3D-files** (description under analyse, chap. 3.1.5) will be automatically performed and two new 3D-files with the same number of x-cuts as well as of y-cuts will be created and loaded. If only the sorting is different it is also possible to first use the option FlipXYsorting3DFile to be found under analyse and then load the second 3D-file.

If the secondary 3D-file has a different startcoordinate in profile direction an automatic shift in profile direction is done if the corresponding message will be confirmed.

The screen is either horizontally or vertically splitted if a second 3D-file is loaded according to the settings within the plotoptions menu (chap. 1.7.6). If the processing option overlay has been activated within the 3D-processing menu the two 3D-files will be overlaid. The parameter second 3D-file under view controls if the secondary 3D-file is shown or not.

Generate 3D-file from 2D-lines: Use this option to generate a 3D-file from existing 2D-lines. After having activated the option a new window appears where you must enter the wanted parameters (see Generate 3D-file from 2D-lines, chap.

3.5). The 3D-data are completely loaded into the RAM of the computer whereby a fast visualization of the data is possible.

Import: import data of different formats into the REFLEX-internal format (see also Import Menu).

With the suboption **3D-file** activated the conversion sequence is automatically set to combine lines/shots (see also Import ConversionMode).

enter Scan3D: enters the module Reflex 3D-Scan which allows to import and to analyse automatically rectangular 3-dimensional GPR-data which have been acquired along 2D-parallel lines in one or two perpendicular directions. Precondition is that the data have been acquired along equidistant parallel 2D-lines on a regular rectangular grid. This means that the traceincrement in one direction (x or y), the startposition of the 2D-lines and the scan increment between the 2D-lines must be equal. The data may have been stored on individual 2D-files or within a 3D-file. If stored within a 3D-file the end-positions of the internal 2D-lines must be identical in addition as well as the number of traces into profile direction. A detailed description of the module is found within chap. 3.1.1.1.

Open several 2D-files: this option allows you to load several (up to 25) different 2D-files in order to display them within the scroll or windows mode. The scale of the different 2D-files can vary. The data can be loaded from a file choice menu or interactively from the profile map (see also interactive choice).

Generate single Timeslices: This option offers the possibility to create so-called time slices, i.e. time constant cuts from a sequence of profiles. The profiles have to be recorded within the same plane. Another precondition is that a static correction to one level was performed before. There are no other prerequisites. The profiles within the plane can be oriented in any direction. The program refers to the corresponding coordinate specifications. The interpolation ranges are of your free choice. Also a time range can be specified over which an averaging is performed. Such an averaging often is necessary because of uncomplete static correction. The details of computing time slices can be found in Generate single timeslices (chap. 3.6).

CopyToClipboard/File: copy the current windows into the clipboard (suboption clipboard) or into a file (suboption file). Different file-formats like BMP, JPEG or TIFF are supported. A clipboard scale factor is queried. A factor larger than 1 allows you to increase the size of the bitmap to be transferred and therefore to increase the resolution.

Image1ToClipboard: copy the primary window into the clipboard.

Print: Use this option if you want to print out the data.

With the option **scroll** activated either the current 2D cut (2D-line) or the total number of 2D cuts (2D-lines) currently chosen within the option box (see Scroll 3D-file) are printed (see also option print several files in chapter 1.8.3).

With the option **window** activated the currently chosen 2D-cuts (2D-lines) are printed out on one paper sheet using the current window combination. After having finished the Print Menu the complete size of the print output and a corresponding query (okay or not) are shown.

When printing **several 2D-lines** with the option **print several files** under PrintOptions2 (see also chapter 1.8.3) activated a filechoice query appears where you must select the wanted profiles. The chosen profiles are sorted alphabetically and are printed according to the current window combination. An example shall make the proceeding clear once more: You want to print 20 parallel profiles and for this combine 4 profiles each time in 2*2 windows. First you have to choose 4 arbitrary sample profiles and make the corresponding window selection. In the next step you choose the print button with the option print several files activated and select the desired profiles. The program then automatically prints the first 4 profiles on one paper sheet. After terminating the output the next 4 profiles are taken and processed in the same way.

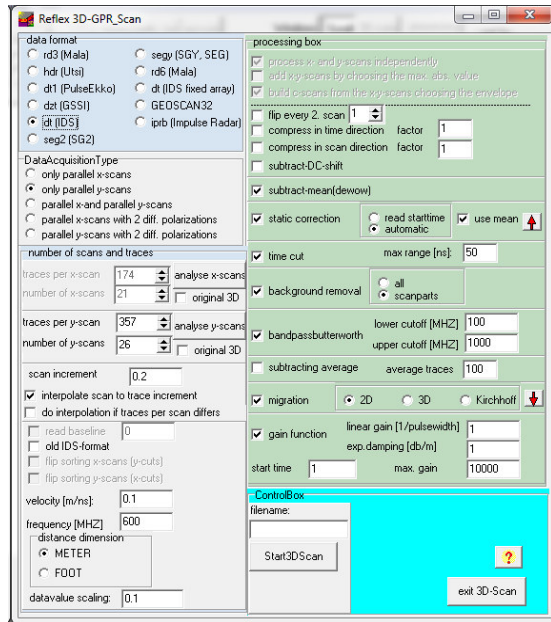
With **print filecomment** activated both the comment stored in the fileheader of the 3D-file and an additional comment about the current position of the 2D-cut are printed out. See also Print Menu.

Edit FileHeader: This option allows to display and change the parameters stored in the global file header and the individual trace headers. These parameters are for example sample and trace increment, presetting of coordinates etc. (see also FileHeader Edit).

exit: leave the module 3D-datainterpretation.

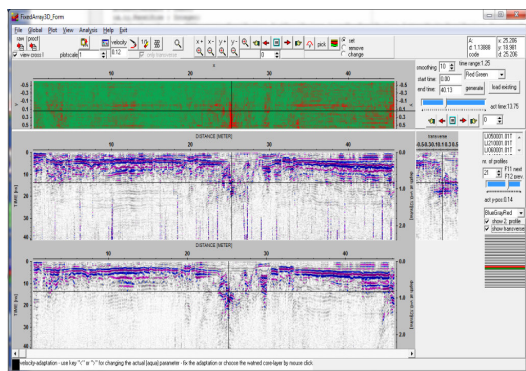
3.1.2 Scan 3D

Enters the Scan 3D tool which allows to import and to analyse automatically rectangular 3-dimensional GPR-data which have been acquired along 2D-parallel lines in one or two perpendicular directions. Precondition is that the data have been acquired along equidistant parallel 2D-lines on a regular rectangular grid. This means that the traceincrement in one direction (x or y), the startposition of the 2D-lines (exception IDS fixed array which allows different start positions) and the scan increment between the 2D-lines must be equal. For a detailed description see chap. 3.10.



3.1.3 3D-fixed array

Enters the 3D-fixed array module which allows the interactive interpretation of multichannel data with a fixed lateral offset of the individual channels based on relative coordinates. The target of this module is the interpretation of data consisting of many traces into one direction, e.g. along a road and a relatively small number of datapoints perpendicular to it (number of parallel lines). An easy and fast calculation of timeslices without interpolation but including smoothing in time direction is incorporated. A fast interactive switch between the different lines and timeslices is possible either using the mouse (pressed right mouse key or using the scroll wheel) or using the trackbars. For a detailed description see chap. 3.9.



3.1.4 View MenuItem

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

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

Picks: activate this option if you want to display picks stored within individual pickfiles for the display of individual 2D-lines. After having activated the option, you must choose the wanted pickfiles from the filelist. Within the fileheader of each pickfile the corresponding datafilename is stored (see also Pick save MenuItem). This filename serves for the assignment of the chosen pickfiles to the chosen 2D-datafiles. To be considered: the datafilename size must not exceed 20 characters including file extension.

fast data cube display: opens the fast data cube display window (chap. 3.7) which allows you a 3D-view of the currently loaded 3D-file. It supports an interactive rotation of the 3D cube. The rotation can be achieved either by mouse interaction or by using scroll bars. The data can be viewed from any direction and can be zoomed. It is possible to bring the fast data cube display window to the front by pressing the right mouse button within the data window of the 3D-datainterpretation.

second 3D-file: allows you to deactivate the display of a second 3D-file - see also Open second 3D-file within the 3D-File MenuItem (chap. 3.1.1).

Overlaid second 3D-file: allows you to deactivate the display of a second 3D-file which has been opened as overlay - see also Open second 3D-file within the 3D-File MenuItem (chap. 3.1.1).

Pick surfaces for 3D-cube: the option allows a 3-dimensional display of the 3D picks within the 3D-cube display (chap. 3.4) together with the data. After having activated the option you are asked for the wanted pickfile(s). An interpolation scheme is used for the picks belonging to one layer.

With the activated option **use code** within the 3D picking panel (chap. 3.8) the pick code stored with the picks is used for discriminating the layers. After having selected the wanted pickfile(s) you are asked for the codes and therefore for the layers to be displayed. The colors of the layers are connected to the pick code colors.

With the option **use code** deactivated the layer number stored with each pickfile is used for discriminating the layers. The colors of the layers are connected to the layershow colors.

After having loaded the wanted pickfiles and after having entered the wanted pickcodes (if option use code is activated) you still must specify the interpolation range in x-, y- and z-direction. Enter a higher value if the pick sampling interval is larger than the data sampling or if the layer boundaries are not completely filled. With the option Pick surfaces for 3D-cube activated the layer boundaries are displayed within the 3D cube display in addition. For example you may combine the back display and the picked layers. The layer display is controlled by the options used for the full plot within the 3D cube display.

Model contours: allows to underlay the model structure which has been used for the FD-simulation. After having activated the option the modelfile will be queried and the contours of the individual layers are shown. The option is only available if a 3D-datafile has been loaded and is only active when displaying snapshots or timeslices. The option is useful for an easier interpretation of the snapshots of a finite difference wavepropagation simulation.

Different models can be loaded for the primary and the secondary 3D-datafile. If only one model has been loaded either using the options primary file or secondary file the model contours of this model will be displayed for both 3D-datafiles.

core data/1D-models: allows to view and adapt 3D-coredata. In addition to the 2D-core data the y-position of the coredata must have been specified in addition within the second line of each core (see example below 28 m).

CORE 4 - NAME4

	30.0000	28.0000		
1	2.500000	1	0.083	0.083
2	1.500000	1	0.087	0.096
3	2.000000	1	0.101	0.146
4	2.000000	1	0.091	0.070

The boreholes will be displayed both within the individual cuts and within the 3D-cube.

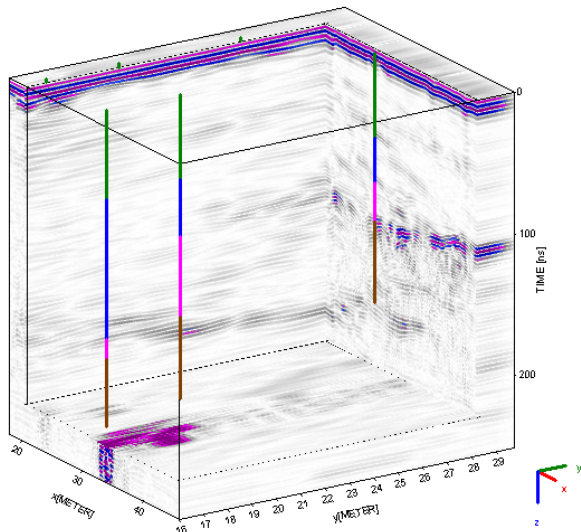
An option box named core adapt opens when a coredata file has been loaded. Within this box the velocities of all layers within the corefile or of a special layer can be adapted. It is only possible to adapt the single velocities of the core within the scroll display mode. Click on any layer bar which you want to adapt. The fill style of this layer bar changes. With the highlighted adaptation parameter v it is now possible to change the velocity of this distinct layer by pressing the key $<$ or $>$ respectively or by entering a new value. With the option **change all** activated all layervelocities of all actual cores are changed simultaneously. This is possible for

all display
modes. The
speedbutton



allows to

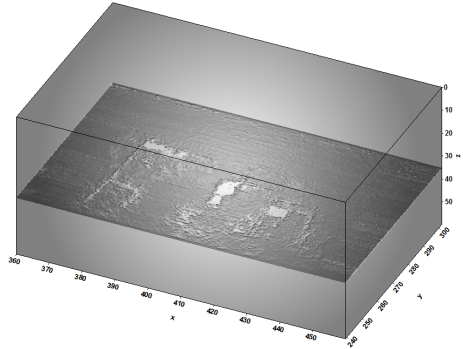
save the adapted
cores
under
the same
filename.



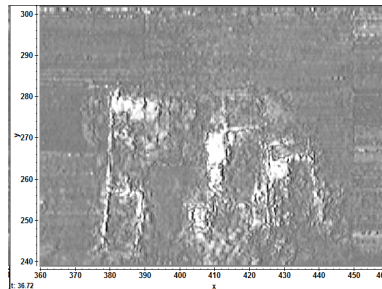
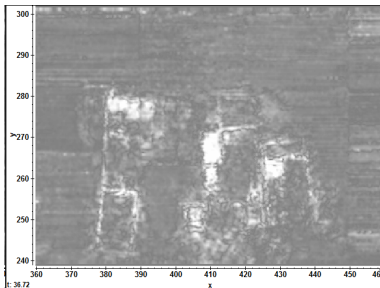
Effects: These display possibilities act both on the scroll and the windows option as well as on the 3D-cube display. The display routines may be applied on all cuts but they are mainly developed for the display of the timeslices. With the option **apply to timeslices only** activated the chosen effects will only be applied to the timeslices and not to the x- and y-cuts. Activating the option opens a new window where the following image effects may be chosen:

bumpmapping: this is a method to simulate lightened rough surfaces by introducing small displacements of the surface. The modified surface normal is then used for lighting calculations. The option intensity (in percent) gives the strength of the modification. The light position may be interactively changed within the blue window using the left mouse button. An example of the bumpmapping for the 3D-cube display is shown on the right.

t: 35.16



Embossing: This is a technique for creating a raised pattern. The lower picture shows an example (left original, right using the embossing method).



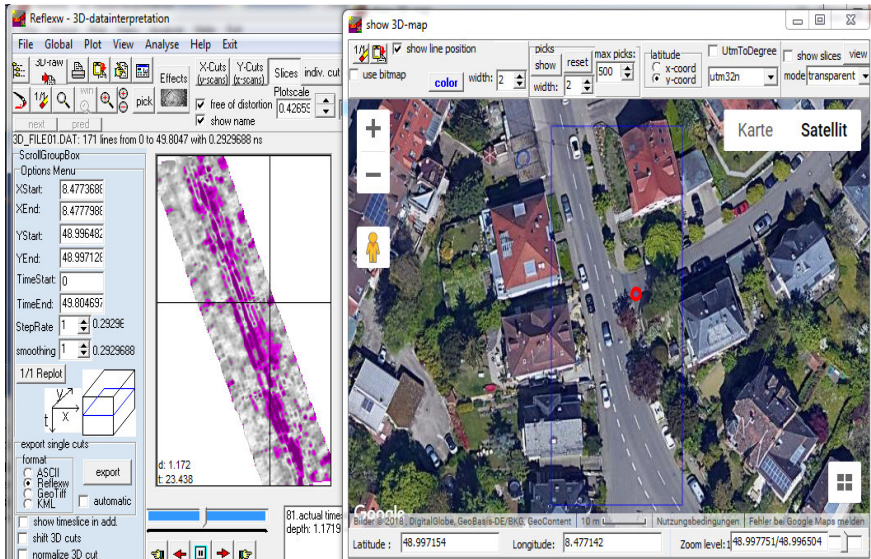
Morphological operations: These methods **median**, **erosion**, **dilation** and a combination of them are the same as given within the Scroll/Windows filter option menu. Here they also act on the 3D-cube display.

The **WallisFilter** is a locally adaptive image enhancement based on Wallis (see Pratt, 1991). This enhancement produces good local contrast throughout the image, while reducing the overall contrast between bright and dark areas. Note: the application of the filter requires quite much computer time. **Blur** applies a Gaussian Blur filter of specified radius defined within the Windows size option.

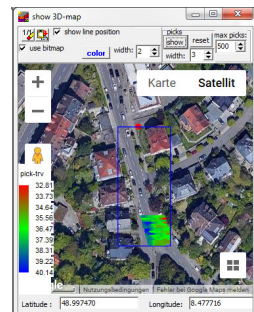
mean max values: if activated the mean and the max.(absolute) values of the actual cut will be displayed within the actual slice memo.

gps map: allows to view the 3D-area within a GPS Map service like OpenLayers, Google, Bing, Azure, TomTom, Here and MapBox.. The default service is OpenLayer. The option **map service** allows to choose any of the other ones. If you want to switch between the different Map services you must have different API keys. Only the OpenLayer service does not need a valid key. If you switch for example to Bing, you should first enter the valid Bing key within the global settings menu if not already done- option **GPS Map API key**. This key will be stored within the Reflexw.ini file.

The **width** and **color** of the rectangle defining the 3D-area may be adapted. Activating the option **show line position** allows to continuously show the actual mouse position by a red circle when dragging over the cut. The option **UTMToDegree** allows to convert UTM-coordinates to degree values based on a given utm zone. The option **latitude** defines where the latitude coordinates have been stored within the traceheaders (x- or y-position). **Zooming** and **copy to clipboard** possibilities are included. The option **1/1** resets and centers the display.

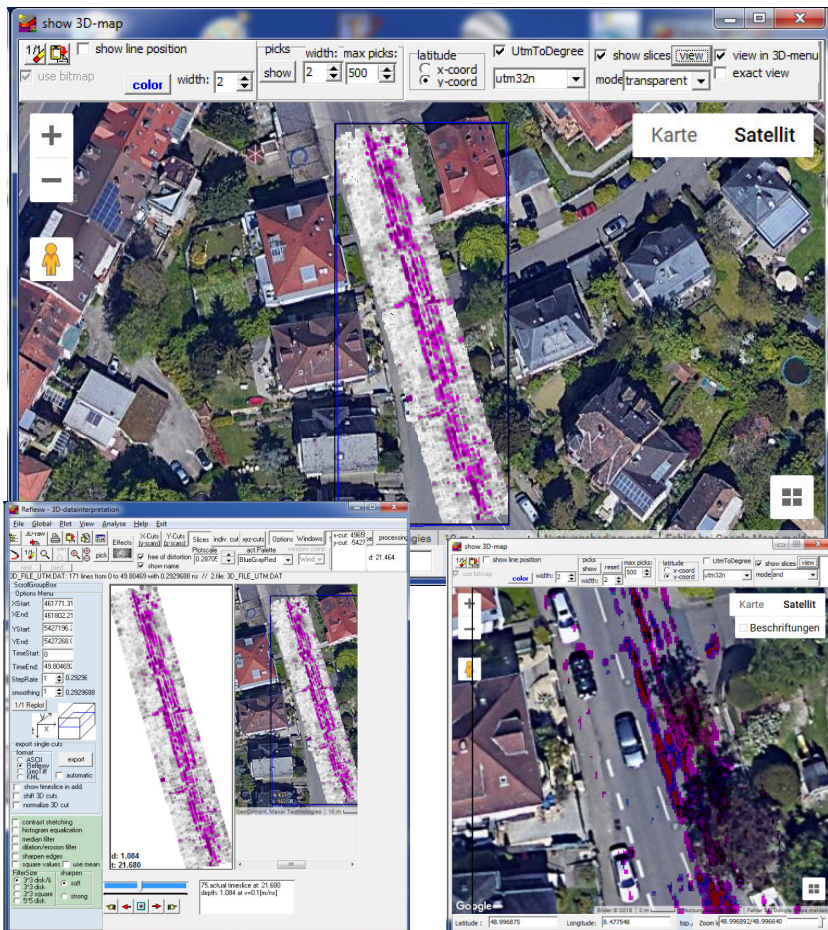


The actual **picks** may be displayed (option **show**) using a red-green-blue color scheme for the icons with red corresponding to the smallest traveltimes values and blue to the largest traveltimes values. The parameter **max. picks** controls the max. picks to be displayed. If the actual number of picks exceeds this number only every n. pick will be displayed. **Reset** resets the pick display. The option **use bitmap** will be automatically activated and the color scheme for the pick traveltimes will be displayed at the left border.



With the option **show slices** activated the actual slice will be displayed within the gps map. The plot option ShowAxis will be automatically deactivated and the option FlipYAxis only slices will be activated. The

the option **mode** allows different combination modes of the gpsmap and the timeslices. With the plotoption **background for 0 value** activated (if not activated a query appears if the option shall be activated) the non covered background may be transparent. With the mode and activated the gpsmap and the timeslice may be overlaid. Other modes are copy, xor and or. The option **view in 3D-menu** allows to view the gps-map including the actual slice within the secondary window of the 3D-datainterpretation menu. The size of the secondary window is fixed but the full gps map will be plotted there and the scroll bars allow to move to the wanted area. With the option **exact view** the slices will not be only copied from the 3D-datainterpretation but will be completely replotted within the gps map menu. This takes much more time and might be useful if a UtmtoDegree transformation is done because in this case a rectangle within utm coordinates is not exactly a rectangle within the degree coordinate system.



3.1.5 Analyse MenuItem

create MPEG-file: option allows to create a MPEG moviefile for the later use with a MPEG-player. The MPEG moviefile may only be created for the Scroll and the Cube3D mode. After having activated the option a new window named create MPEG-file opens which stays on the top. First you must enter the **filename** (without extension). The MPEG-file will be stored under the current projectdirectory with the extension mpg.

The **compression** is controlled by the trackbar and the checkbox **MJPEG**. The trackbar allows you to either increase the **quality** (trackbar to the left) or to increase the **compression** (to the right). With the highest compression the filesize will be decreased by a factor of about 4 compared to the highest quality. With the option MJPEG deactivated an additional encoding is used which again reduces the filesize by about 50 %. In this case the computertime for constructing the MPEG-file will be significantly increased and the quality as well will be decreased.

In the frequency Radio Group box you may set the **frequency**, with which the certain pics of the mpeg-file will be shown (8 Hz means 8 pics per sec, e.g.). Start the recording by pressing the **record** button. Now the other 3 buttons (play, pause and stop) are enabled. The **play** button will be automatically activated and the first slide is added to the MPEG movie. As long as the play button is activated each new slide will be added if the actually shown data is changed (e.g. when using the scroll bars or when using the plot option). The current slide number is shown. If you want to interrupt the MPEG-construction use the button **pause**. The button **stop** finishes the construction and the MPEG-file will be closed.

To be considered: the bitmap size must not be changed. Therefore an error message appears and the MPEG construction will be stopped if the bitmap size, e.g. after having resized the 3D-datainterpretation window, has been changed. The bitmap size is determined when pressing the record button.

interpolate current 3D-file: the option allows a reinterpolation of the current 3D-file into the different directions. After having entered the options you must specify the interpolation factors into the 3 different orientations. The number of points into the individual orientations increases accordingly. For example the option is useful if the data density in one direction is too small for a good 3D-cube display. The option **check zero traces and traceheader** controls if zero traces are present within the 3D-cube and sets those traces again to zero. This is necessary because the interpolation is done using a FastFourierTransform which generates border effects. Activating this option also preserves the traceheader coordinates of the original file. No interpolation for the coordinates is done. The interpolated traces will have the same traceheader coordinates. Activating the option significantly increases the needed processing time.

flipXYsorting 3D-file: the option allows to change the internal sorting of the traces of the current 3D-datafile. A 3D-datafile consists of several parallel 2D-lines which are orientated into either x- or y-direction (defined within the fileheader - profile direction). After having applied this option the parallel 2D-lines of the 3D-datafile are orientated into the other direction (e.g. original x - after the option they orientated into y). The option may be useful in order to compare two different 3D-datafiles which are measured into 2 different directions (crossing orientation) because the program only allows to compare 2 different 3D-datafiles which have the same number of traces/2D-lines and of 2D-lines and which internal sorting is

the same. Activating the option **check zero traces and traceheader** preserves the traceheader coordinates of the original file. No interpolation for the coordinates is done. The interpolated traces will have the same traceheader coordinates. Activating the option significantly increases the needed processing time.

Shift 3D-cuts: allows to shift the cuts against each other (see also chap. 3.2 scroll 3D-file - option shift 3D-cuts)

Normalize 3D-cuts: allows an energy normalization for up to 10 different rectangular xy-ranges (see also chap. 3.2 scroll 3D-file - option normalize 3D-cuts).

adjust 2 3D-files: The option allows to adjust two 3D-files which have a different number of x- and/or y-cuts for a later comparative analysis (for example one 3D-file with only x-cuts and another 3D-file with only y-cuts). After having chosen the wanted two 3D-files using the option **select two 3D-files for adjusting the program** automatically calculates the min./max. xy-range and a new rasterincrement based on the profile increment (the profile increment must be a multiple of the new rasterincrement). The original increments and the min./max. xy-ranges of the two 3D-files are listed within the two memo windows. After having activated the option **start** the following steps will be performed:

- traceincr-resampling based on the new rasterincrement
- expand 3D-file using the same rasterincrement in x- and y-direction
- include zero traces for the xy-range which has not been covered by the original data
- flipXYsorting of one 3D-file if the two original 3D-datafiles have been acquired on crossing lines.

With activated option **update traceheader coordinates** the traceheader coordinates of the new 3D-files will be automatically updated.

The filenames of the new 3D-files is the same like the original files but with a different extension based on the entered processing label and the files will be stored under the path procddata. As a result two 3D-files with the same number of x-cuts as well as of y-cuts will be created. In addition the rasterincrements in x- and y-direction are identical.

adjust two 3D-files

processing label: 0

select two 3D-files for adjusting

F:\data\ids_schuler_array\XZ_FILES_01.DAT
 xmin: 0 / xmax: 6.570239916
 ymin: 0 / ymax: 5.75
 xincr: 0.02783999965 / yincr: 0.25

F:\data\ids_schuler_array\YZ_FILES_01.DAT
 xmin: 0.5 / xmax: 6
 ymin: 0 / ymax: 6.013439924
 xincr: 0.25 / yincr: 0.02783999965

3D coordinates:

XStart: 0
 XEnd: 6.570239916
 YStart: 0
 YEnd: 6.013439924
 Rasterincrement: 0.027777777

☒ update traceheader coordinates

Start Cancel

data are loaded into RAM

The option adjust 2 3D-files will be automatically performed if a **secondary 3D-file** shall be loaded (option file/open second 3D-file) and the number of x- and y-cuts do not fit.

3.1.6 3D-file processing

Within this menu you may enter the type of display for both the primary and secondary 3D-file.

no means that the data are displayed as they are.

absoluteValues: if activated the absolute values are calculated and shown. Activating this option is useful for slices for example.

envelope: if activated the envelope of each trace of the 3D-file is calculated and shown. Activating this option is useful for slices for example.

envelope timeslices only: with this option activated the envelope of each trace of the 3D-file is calculated and shown for the timeslices only (option slice activated). For the other cuts (x-cuts and y-cuts) the “normal” traces will be shown. The option is not available for a secondary 3D-file and will also be automatically deactivated for the primary 3D-file if a secondary 3D-file has been loaded.

overlay: The option allows to overlay the secondary 3D-file. The chosen processing option for the primary 3D-file controls how the overlaying is done. With the option no or abs. amplitude activated the two 3D-files are simply added for all 3 cuts. The option always max.value for overlay (given within the panel for the secondary 3D-file) controls whether the summing is done based on the max.value (option activated) or based on an averaging over the crossing points (option deactivated). With the option envelope and envelope timeslices only always the max. value is used for overlaying all cuts for which the envelope has been created. The option overlay might be useful for a combined interpretation of two 3D-files with different antenna polarization (GPR-data) or with different orientation of the original 2D-lines.

For timeslices only: The option allows to use the secondary 3D-file only for the display of the timeslices. For the display of the x- and y-cuts the primary file will be used. This might be useful if you use a nonmigrated 3D-dataset as primary file and the migrated 3D-dataset as secondary 3D-file. In this case the migrated dataset will be used for the display of the timeslices and for the x- and y-cut display the non migrated data are used which may allow a better detection of small objects in the form of diffraction hyperbola in some cases.

With the option **square** activated the data are squared when loading. The sign will be preserved and the resulting amplitudes will be normalized to the original maximum value. With the option **use mean** activated the normalization will be done based on the mean value. Using the option square allows a higher contrast. High amplitude objects will be emphasized in comparison to small amplitude reflections/ diffractions. Use this option instead of the scroll and window 2D-filter option square if you also want to use the 3D-cube display with the squared amplitudes.

With the option **envelope timeslices only** activated there exist some **2D-filter** which will be applied onto the timeslices. These filters correspond to those given for the scroll option (chap. 3.2) but in contrast to them these filters also act for the other display modes windows and 3D-cube.

contrast stretching: if activated a contrast stretching will be applied. Contrast stretching is a simple image enhancement technique that attempts to improve the contrast in the image by 'stretching' the range of intensity values it contains to span a desired range of values. It differs from the more sophisticated histogram equalization in that it can only apply a linear scaling function to the image pixel values. As a result the 'enhancement' is less harsh. If activated the function drastically changes the amplitude distribution within the slice. Therefore no significant statement over the original amplitude distribution is possible any more.

histogram equalization: if activated a histogram equalization will be applied. The histogram shows the occurrences of each intensity value in the image. Histogram equalization is a technique by which the dynamic range of the histogram of the image is increased. Histogram equalization assigns the intensity values of pixels in the input image such that the output image contains a uniform distribution of intensities. It improves contrast and the goal of histogram equalization is to obtain a uniform histogram. If activated the function drastically changes the amplitude distribution within the slice. Therefore no significant statement over the original amplitude distribution is possible any more.

median filter: The median filter smooths smallscale heterogeneities but preserves edges. The filter size must be entered. 3*3 disk/li. and 3*3 disk includes 5 points, 3*3 square 9 points and 5*5 disk 13 points.

dilation/erosion filter. Dilation and erosion are morphology-based operations. Dilation, in general, causes objects to dilate or grow in size but also a hole fitting will be observed. Erosion causes objects to shrink. The combination of both has a similar effect like the median filter (see picture on the right, left panel original, right panel effect of dilation/erosion filter).

The filter size must be entered. 3*3 disk/li. and 3*3 disk includes 5 points, 3*3 square 9 points and 5*5 disk 13 points. The filter effect increases with the filtersize (3*3 disk/lin. has the smallest effect).

The filter is mainly useful for the display of slices.

sharpen edges: if activated an algorithm will be applied for the plotting which sharpens the edges of the different elements within each image. The sharpen strenghts soft and strong can be entered.

3.2 Scroll 3D-file

Using the option "scroll" you may continuously move through the 3D-cube either in x-, y- or z-direction using the track bar. If you have loaded several 2D-lines (option Open several 2D-files - see 3D-File Menuitem) you may continuously move through the set of 2D-lines. The step and smoothing rate is freely selectable.

The control panel below the track bar allows an automatic scrolling through the 3D-cube:



scroll to smaller cuts



stop scrolling



scroll to higher cuts

Use the options **X-Cuts**, **Y-Cuts** and **Slices** to determine which 2D-cut of the 3D-file you want to be shown. With X-Cuts activated all existing cuts perpendicular to the x-direction may be chosen. With Y-Cuts activated all existing 2D-cuts perpendicular to the y-direction may be chosen. With Slices activated all existing 2D-cuts along the time-direction may be chosen. The current 2D-cut within the total 3D-cube (a 3D-file has been loaded) is shown within the lower image of the option menu.



replot current 2D-cut with current zoom parameters



resets the x- and y-scale values (zoomvalues) to 1 and replots the 2D-cut



enable magnifying glass function



enable manual zoom - With the option ZOOM an arbitrary area of the 2D-cut can be selected and plotted in full screen size. The area to be enlarged, a rectangle, has to lie within a data set. Pressing the left mouse button you determine a corner of this rectangle and by moving the mouse with pressed button the desired area is opened. The zoom range may be changed step by step using the small + or - buttons on the right side of the manual zoom button. Clicking on the + button increases the zoom by 10 %, clicking on the - button decreases the zoom by 10 %, this means the x-y-range to be shown will be larger. For scrolling use the left and lower scroll bars.

If **Options** is activated the option panel at the left allows you to specify some display parameters:

With the option **show name** activated the filename or the current slice is shown at the top or bottom (flip y axis active) of each cut (line).

free of distortion: activate this option if you want the display of the slices to be free of distortion, this means the axis in x- and y-direction have the same scale. The option is only active for slices.

The first two **start and end parameters** specify the range to be displayed from left to right. With X-Cuts activated this normally means the y-direction. With Y-Cuts activated this normally means the x-direction. With Slices activated the parameters specify the x-direction.

The following two parameters specify the range to be displayed from top to bottom. With X-Cuts or Y-Cuts activated this normally means the time range. With Slices activated the parameters specify the y-direction.

The next two parameters specify the range in scroll direction. With Slices activated these parameters specify the wanted time range for example.

StepRate: this parameter allows you to change step rate for scrolling. Every step range 2D-cut will be plotted when scrolling.

smoothing: enter a value for smoothing the data in the scroll direction. A value of 1 means no smoothing. A value of 2 means that 2D-cuts are summed up and the mean values are calculated. The smoothing parameter is independent from the StepRate parameter. The smoothing is done symmetrically in both directions (half the smoothing value in each direction). With the option **use max.value** activated only the max. value will be used within the scroll range, otherwise the mean value will be calculated.

Plot: replot the current 2D-cut.

ShowTimeSliceInAddition: if activated the current timeslice depending on the mouse position is shown in a second window for the scroll mode. The timeslice is shown based on the envelope data. The current cursor position is marked by a cross within the timeslice. The screen is either split vertically or horizontally depending on the settings within the plotoptions. The option will be automatically deactivated if one switches to the windows or the 3D-cube mode. The option is also not available if "envelope timeslices only" has been chosen within the 3D-processing menu. The left mouse button allows to scroll one cut forwards and the right mouse button to scroll backward (precondition: pick option deactivated). If the cut option slices has been activated the x- and y-cuts corresponding to the actual mouse position within the timeslice will be shown within the second window.

The same holds true if the 3D-file is a FD-snapshot file x-line. In this case the timeslice is replaced by the cut option snapshot.

There exist some **2D-filter** which will be applied onto the actual cuts if activated:

contrast stretching: if activated a contrast stretching will be applied. Contrast stretching is a simple image enhancement technique that attempts to improve the contrast in the image by 'stretching' the range of intensity values it contains to span a desired range of values. It differs from the more sophisticated histogram equalization in that it can only apply a linear scaling function to the image pixel values. As a result the 'enhancement' is less harsh. If activated the function drastically changes the amplitude distribution within the slice. Therefore no significant statement over the original amplitude distribution is possible any more.

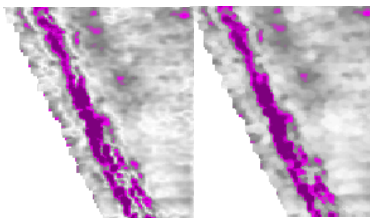
histogram equalization: if a activated a histogram equalization will be applied. The histogram shows the occurrences of each intensity value in the image. Histogram equalization is a technique by which the dynamic range of the histogram of the image is increased. Histogram equalization assigns the intensity values of pixels in the input image such that the output image contains a uniform distribution of intensities. It improves contrast and the goal of histogram equalization is to obtain a uniform histogram. If activated the function drastically changes the amplitude distribution within the slice. Therefore no significant statement over the original amplitude distribution is possible any more.

median filter: The median filter smooths smallscale heterogeneities but preserves edges. The filter size must be entered. 3*3 disk/li. and 3*3 disk includes 5 points, 3*3 square 9 points and 5*5 disk 13 points.

dilation/erosion filter. Dilation and erosion are morphology-based operations. Dilation, in general, causes objects to dilate or grow in size but also a hole fitting will be observed. Erosion causes objects to shrink. The combination of both has a similar effect like the median filter (see picture on the right, left panel original, right panel effect of dilation/erosion filter).

The filter size must be entered. 3*3 disk/li. and 3*3 disk includes 5 points, 3*3 square 9 points and 5*5 disk 13 points. The filter effect increases with the filtersize (3*3 disk/lin. has the smallest effect). The filter is mainly useful for the display of slices.

sharpen edges: if activated an algorithm will be applied for the plotting which sharpens the edges of the different elements within each image. The sharpen strenghts soft and strong can be entered.

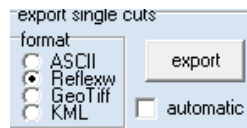


square: if activated the data are squared when loading. The sign will be preserved and the resulting amplitudes will be normalized to the original maximum value. With the option

use mean activated the normalization will be done based on the mean value. Using the option square allows a higher contrast. High amplitude objects will be emphasised in comparison to small amplitude reflections/ diffractions.

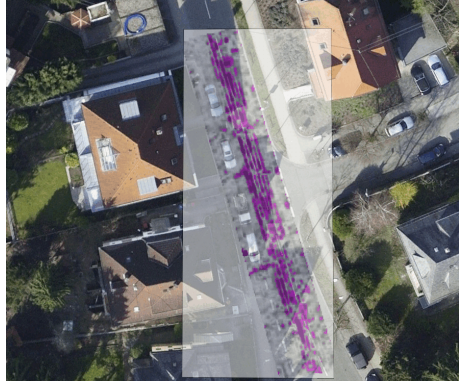
There exist several possibilities to **output** the actual cuts: ASCII, Reflexw, GeoTiff and KML. The option **export** allows an export of the actual cut into the chosen format.

ASCII-export: allows to export the current cut to ASCII GRD-format (see also Dataexport, chap. 1.6). The data are written out sample after sample (option x-(distance)-direction activated within chap. 1.6). The second line contains the number of traces and of samples. The third line contains the start and end-position in x-direction, the fourth line the start and end-position in y-direction. Each of the following lines contains all amplitude values of one y-sample. The option is only enabled within the scroll mode. The ASCII-filename is automatically determined from the 3D-datafilename and the current cut. Example:



3D-datafilename: file01_3D.00t
 current cut: x: 0.58
 resulting ASCII-filename: file01_3Dx_0_58.grd

GeoTiff export: allows to export the current cut to a tiff file together with a tfw world file. A world file file is a plain ASCII text file consisting of six values separated by newlines. The format is:
 pixel X size
 rotation about the Y axis (usually 0.0)
 rotation about the X axis (usually 0.0)
 negative pixel Y size
 X coordinate of upper left pixel center
 Y coordinate of upper left pixel center



The tiff file size corresponds to the actual window size and the following plotoptions are automatically used:
 no axis (plotoption showaxis automatically deactivated)
 max. y-value on top (plotoption flip y-axis automatically activated)
 A clipboard scale factor is queried. A factor larger than 1 allows you to increase the size of the bitmap to be transferred and therefore to increase the resolution. The filename is automatically determined from the 3D-datafilename and the current cut. Example:
 3D-datafilename: file01_3D.00t
 current cut: sclice: 0.58
 resulting tiff-filename: file01_3Dt_0_58.tif
 resulting word file: file01_3Dt_0_58.tfw

Reflexw 2D-file: generates a Reflex 2D-file of the actual cut. The filename is automatically determined from the 3D-datafilename and the current cut and will be stored under the path rohdata under the actual projectdirectory. Example:
 3D-datafilename: file01_3D.00t
 current cut: x: 0.58
 resulting Reflex 2D-filename: file01_3Dx_0_58.dat

KML export: allows to export the current timeslice to a KML file for a later use within Google Earth. Two different creating types are supported:
image overlay: using this type the complete timeslice image will be exported to a png-file. A clipboard scale factor is queried. A factor larger than 1 allows you to increase the size of the bitmap to be transferred and therefore to increase the resolution.

In addition a KML file will be created including the north/south and east/west corner coordinates and the name of the png-file. The original coordinates can be given in degrees or in utm-coordinates. In the second case the option UTMToDegree must be activated and the corresponding UTM-zone must be entered. The advantage of this type is the small resulting KML filesize. The disadvantage is that all data including the non measured area will be displayed.

Create single icons: using this type each point of the timeslice will be exported as a single icon including the coordinates and the colour corresponding to the actual colour settings. The icon size can be changed. The original coordinates can be given in degrees or in utm-coordinates. In the second case the option UTMToDegree must be activated and the corresponding UTM-zone must be entered. Activating the option ignore zero values zero values (e.g. data from the non measured area) will not be taken into account. The disadvantage of this method is the huge resulting KML filesize and that the icon size is independent from the Google Earth view point.

In both creating types the KML filename is automatically determined from the 3D-datafilename and the current cut. Example:

3D-datafilename: file01_3D.00t

current cut: sclice: 0.58

resulting kml-filename: file01_3Dt_0_58.kml

resulting png-filename: file01_3Dt_0_58.png

automatic: With this option activated the export is always done if a new cut will be displayed,. For this possibility the formats ASCII, Reflexw and PNG are supported. For example using the automatic scroll arrows a complete cut set in one direction can be exported within one single step. No scaling factor for the PNG bitmap export will be asked - therefore the resulting bitmap file has the same size as the screen size. The filename is automatically determined from the 3D-datafilename and the current cut and will be saved under the path ASCII under the actual project directory. Example for the PNG export:

3D-datafilename: file01_3D.00t

current cut: sclice: 0.58

resulting png-filename: file01_3Dt_0_58.png



shift-3d-cuts: allows to shift the cuts against each other. The option might be useful if the cuts exhibit a special zig-zag pattern (shift mode set to shift every 2. cut) due to e.g. a meandering data acquisition (see picture on the right) or an overall shift of special xy-ranges (shift mode set to shift every cut). The zig-zag pattern can only be removed if no interpolation has been used when defining the 3D-datafile.

It is possible to define up to 100 different rectangular xy-ranges (parameter number) whereby it is only possible to restrict the range perpendicular to the direction of the acquired profiles. The shift is only possible in the direction of the acquired profiles (not perpendicular to them). The shift will be displayed interactively as soon as any change of the parameters has been done. For that purpose the data should be displayed using the slices. It is possible to save the new 3D-datafile using the option analyse/shift-3D-cuts.

Input parameters:


number: number of the xy-range


Start x-cut or start y-cut: first cut number for the actual xy-range

end x-cut or y-cut: last cut number for the actual xy-range

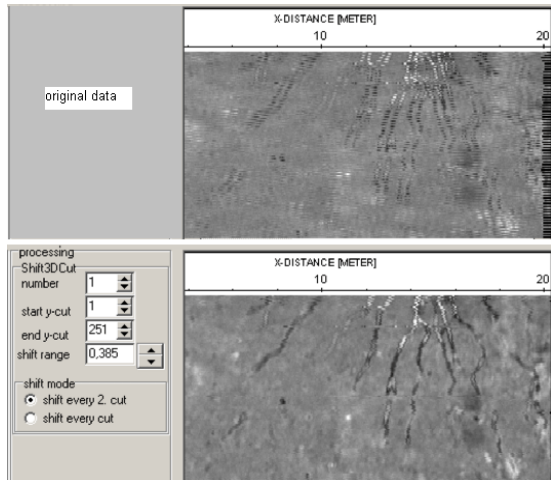
Shift range: input of the shift range using the given distance dimension.

shift mode: enter whether every 2. cut or every cut shall be shifted starting at start cut respectively.

The actual shift parameters can be saved on a file shift3DCuts.fil under the project directory using the option . Those parameters can also be reloaded

using the option .

Using the option analyse/Shift 3D-cuts a new 3D-datafile containing the shifted cuts can be saved. For that purpose a processing label must be entered. The option is equivalent to the processing option Shift-3DFile under processing/trace interpolation within the 2D-dataanalysis which however uses the complete xy-range for the entered shift parameters.



Normalize 3d-cuts: allows an energy normalization for up to 10 different rectangular xy-ranges. The option might be reasonably employed if e.g. the timeslices show distinct stripe patterns which indicate different coupling conditions (see picture on the right). The energy normalization will be displayed interactively as soon as any change of the parameters has been done. For that purpose the data should be displayed using the slices.

Input paramters:

number: enter the number (max. 10) of the xy-range to be normalized


The xy-range is defined within the editfields **min.** and **max.**.

The z-range has been set to the max. range by default. It can be decreased for the normalization based on the constant normalizing factor.

Set to max: set the x/y cuts to the max. values for the actual 3D-profile

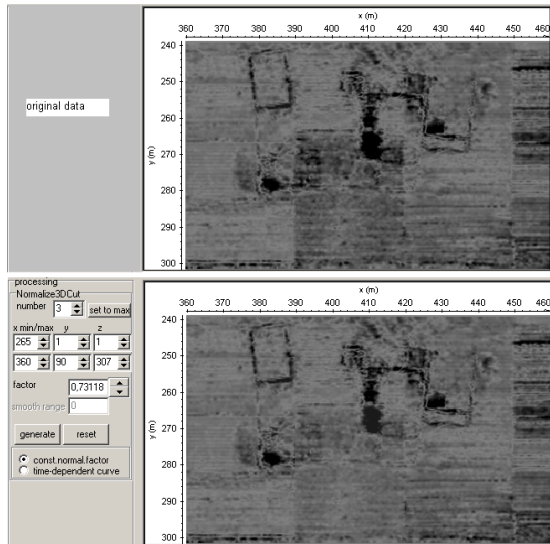
constant normalizing factor: one single scaling **factor** is used for all values lying within the chosen xy-range. Activating the option **generate** the scaling factor will be calculated from the quotient of the overall mean energy value of the 3D-datafile and the total energy value of the chosen xy-range. The mean energy values will be determined within the given timewindow (options min/max. z cuts). The factor can also be be entered (changed) manually.

time-dependent normalizing curve: a timedependent scaling curve will be calculated from the overall mean envelope curve of the complete 3D-file and the mean envelope curve of the chosen xy-range using the option **generate**. The envelope curves will be determined over the total timerange and summed up over all traces. Then this mean envelope (energy) curve will be smoothed over the given timerange (option **smooth range**). The option changes the energy distribution with respect to the timeaxis. For example: if a high reflection energy area exists within a special timerange within the chosen xy-range the option will lower the energy within this timerange for this range.

The actual normalize parameters can be saved on a file `normalize3DCuts.fil` under the project directory using the option .

reloaded using the option .

It is possible to save the new 3D-datafile containing the actual `normalize3DCuts` changes using the option `analyse/normalize-3D-cuts`.

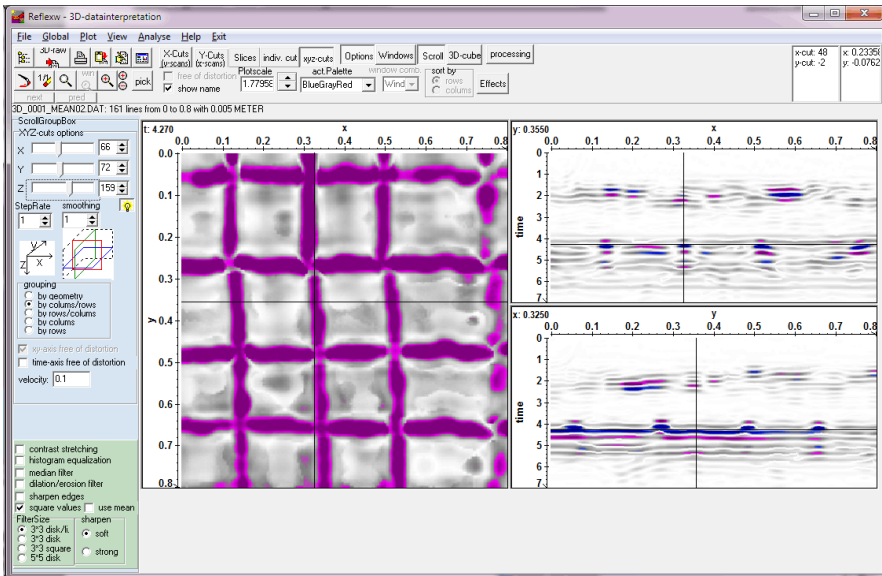


3.2.1 xyz-cuts

The option **xyz-cuts** allows different simultaneous display possibilities of the 3 cuts. The actual positions of the other cuts are indicated by a large cross within each cut.

A fast interactive change of the cuts is done by several mouse actions:

- the mouse wheel can be used in order to go through any of the cuts
- pressing the left mouse button allows to control the other cuts within any cut.



The **grouping** box allows the following display types:

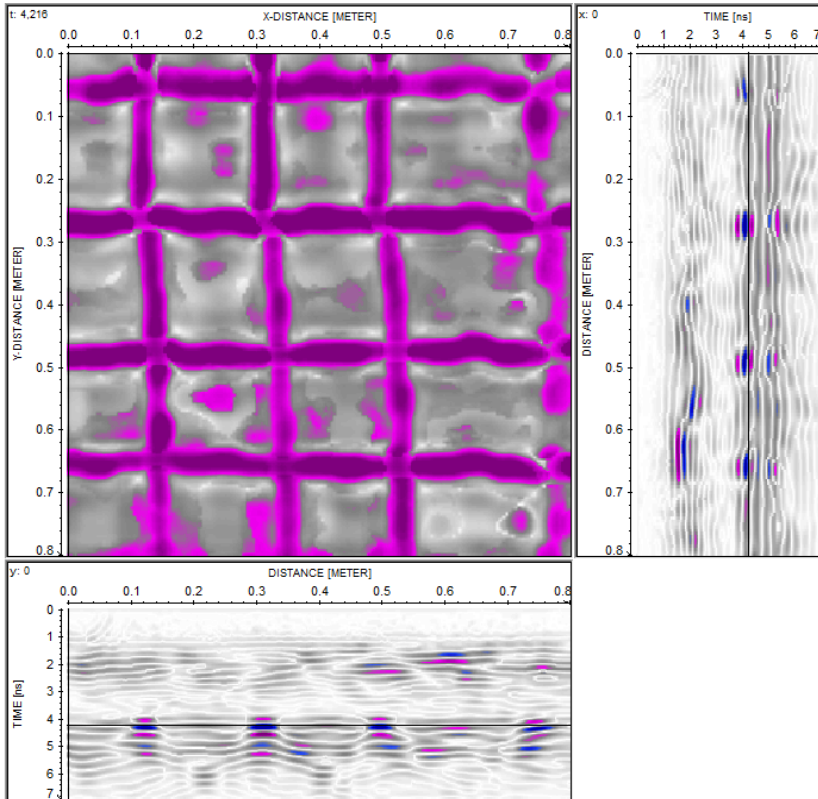
- **by geometry** (see picture below). The z-cut (slice) is displayed on the top/left surrounded by the x- and y-cut using a free of distortion scale. The time axis scale of the xy-cuts may be adjusted to the xy-scale activating the option **time axis free of distortion**.
- **by columns/rows** (see picture above). The xy-cuts are displayed right to the z-cut (slice). The time axis scale of the xy-cuts may be adjusted to the xy-scale activating the option **time axis free of distortion**.
- **by rows/columns** (see picture below). The xy-cuts are displayed below the z-cut (slice). The time axis scale of the xy-cuts may be adjusted to the xy-scale activating the option **time axis free of distortion**.
- **by columns** (see picture below): all three cuts will be displayed within one row with the z-cut (slice) in the middle
- **by rows** (see picture below): all three cuts will be displayed within one column with the z-cut (slice) in the middle

If the option **only slices** for the smoothing is activated the smooth factor only acts on the slices and not on the x- and y-cuts.

Two options control the scale:

xy-cuts free of distortion: if activated the xy-scales are the same. The option is always activated for the grouping “by geometry” and “by columns/rows”.

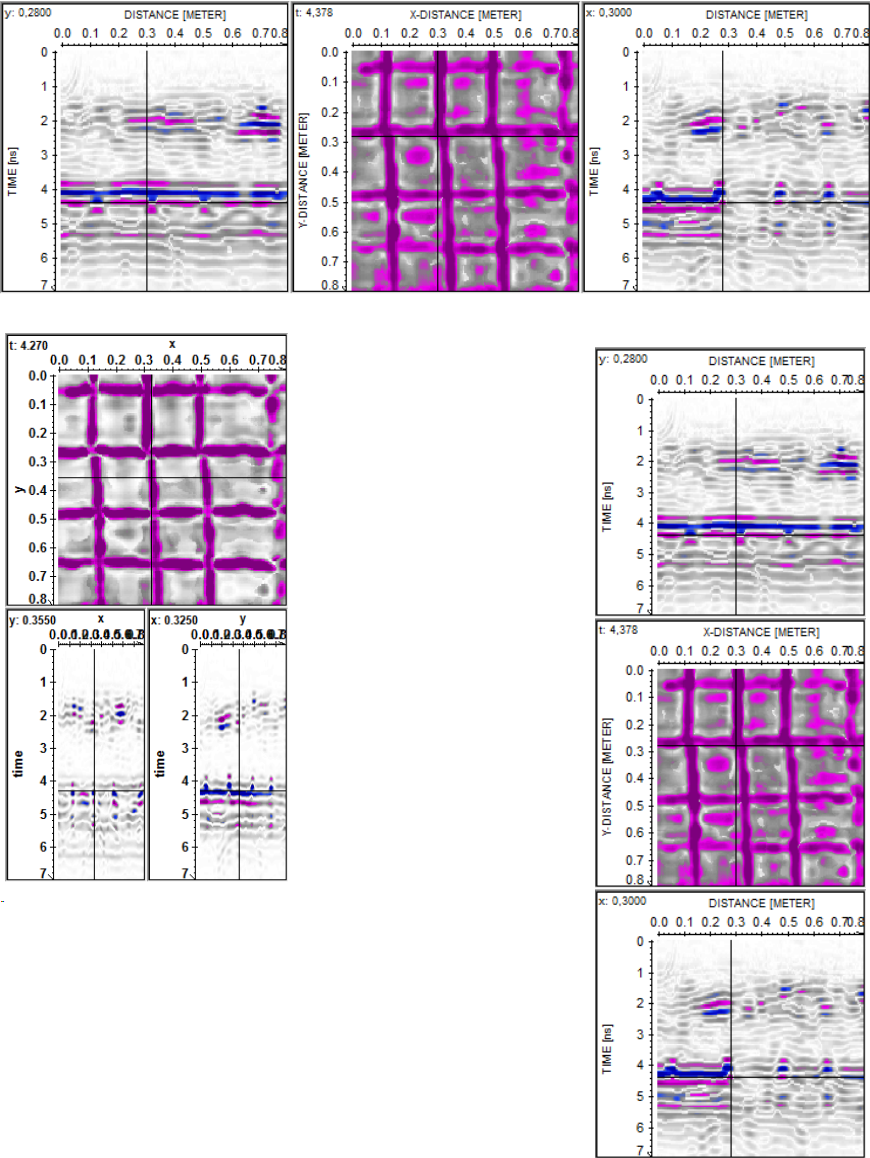
time axis free of distortion: if activated the time axis scale is calculated from the timerange and the velocity entered for the depth axis within the plotoptions menu. The option should be activated/deactivated for the grouping “by geometry” if the axis scales do not fit.



It is possible to change the point of view within all 3 cuts. A large cross shown within each cut defines the actual positions of the other 2 cuts. In addition a 3D-sketch is shown within the option panel. Moving the mouse with pressed left button allows to change the view position of the other 2 cuts. Using the mouse scroll bar allows to go through the actual cut. With the yellow light option activated the cut automatically gets the focus when entering it. Otherwise the wanted trackbar (X, Y or Z) must be chosen or the left mouse button must be clicked within the wanted cut and the corresponding cut will get the focus.

Pecularity for **picking**: setting picks is only possible using the right mouse button.





3.3 Windowing 3D-file

Using the option "windows" you may display several 2D-cuts of the 3D-cube or several individual 2D-lines, resp., placed in a freely selectable manner within the working window. The maximum number of individual images is 25.

The step and smoothing rate may be freely chosen. The program automatically determines the StepRate from the existing number of 2D-cuts and the maximum number of windows using the following formula: number of 2D-cuts / 25.

Use the options **X-Cuts**, **Y-Cuts** and **Slices** to determine which 2D-cut of the 3D-file you want to be shown. With X-Cuts activated all existing cuts perpendicular to the x-direction may be chosen. With Y-Cuts activated all existing 2D-cuts perpendicular to the y-direction may be chosen. With Slices activated all existing 2D-cuts along the time-direction may be chosen. The current 2D-cuts within the total 3D-cube (a 3D-file has been loaded) are shown within the lower image of the option menu.



replot current 2D-cuts with current zoom parameters



resets the x- and y-scale values (zoomvalues) to 1 and replots the 2D-cuts



enable magnifying glass function



with win: magnifying glass function for the complete data-window the cursor is placed. The size of the magnifying glass window is automatically determined from the current zoomfactor and the size of the data-window.



enable manual zoom - With the option ZOOM an arbitrary area of the 2D-cuts can be selected and plotted in full screen size. The area to be enlarged, a rectangle, has to lie within one single 2D-cut. Pressing the left mouse button you determine a corner of this rectangle and by moving the mouse with pressed button the desired area is opened. Scrolling is not possible within the Windows-mode.

The option **window combinations** allows you to choose the combination of the windows to be subdivided in horizontal and vertical direction. If you have 20 different cuts to be displayed, you may choose the combination 4/5 - this means the screen is subdivided into 4 horizontal and 5 vertical cuts.

The options **sort by rows** and **sort by columns** define the sorting. With sort by rows activated the first cuts are displayed within the first row, the next ones within the second row and so on. With sort by columns activated the first cuts are displayed within the first column, the next ones within the second column and so on.

The radio buttons **sort by filenames**, **sort by const.coord.** and **sort by prof.coord.** specify how the several individual 2D-files are sorted for the display. If you have loaded a 3d-file, the options are disabled. With **sort by filenames**

activated the 2D-files are sorted by files with ascending order. The chosen lines are alphabetically resorted (sorted with ascending order - see also Generate 3D-file from 2D-lines). With **sort by const.coord.** activated the 2D-files are sorted by the coordinates in profile constant direction with ascending order. With **sort by prof.coord.** activated the 2D-files are sorted by the coordinates in profile direction with ascending order.

If **Options** is activated the option panel at the left allows you to specify some display parameters:

With the option **show name** activated the filename or the current slice is shown at the top or bottom (flip y axis active) of each cut (line).

free of distortion: activate this option if you want the display of the slices to be free of distortion, this means the axis in x- and y-direction have the same scale. The option is only active for slices.

The first two **start and end parameters** specify the range to be displayed from left to right. With X-Cuts activated this normally means the y-direction. With Y-Cuts activated this normally means the x-direction. With Slices activated the parameters specify the x-direction.

The following two parameters specify the range to be displayed from top to bottom. With X-Cuts or Y-Cuts activated this normally means the time range. With Slices activated the parameters specify the y-direction.

The next two parameters specify the range in scroll direction. With Slices activated these parameter specify the wanted time range for example.

StepRate: this parameter allows you to change step rate for cutting. Every step range 2D-cut will be plotted.

smoothing: enter a value for smoothing the data in the cutting direction. A value of 1 means no smoothing. A value of 2 means that 2D-cuts are summed up and the mean values are calculated. The smoothing parameter is independent from the StepRate parameter. The smoothing is done symmetrically in both directions (half the smoothing value in each direction).

Plot: replot the current 2D-cuts.

There exist some **2D-filter** which will be applied onto the actual cuts if activated:

contrast stretching: if activated a contrast stretching will be applied. Contrast stretching is a simple image enhancement technique that attempts to improve the contrast in the image by 'stretching' the range of intensity values it contains to span a desired range of values. It differs from the more sophisticated histogram equalization in that it can only apply a linear scaling function to the image pixel values. As a result the 'enhancement' is less harsh. If activated the function drastically changes the amplitude distribution within the slice. Therefore no significant statement over the original amplitude distribution is possible any more.

histogram equalization: if a activated a histogram equalization will be applied. The histogram shows the occurrences of each intensity value in the image. Histogram equalization is a technique by which the dynamic range of the histogram of the image is increased. Histogram equalization assigns the intensity values of pixels in the input image such that the output image contains a uniform distribution of intensities. It improves contrast and the goal of histogram equalization is to obtain a uniform histogram. If activated the function drastically changes the amplitude distribution within the slice. Therefore no significant statement over the original amplitude distribution is possible any more.

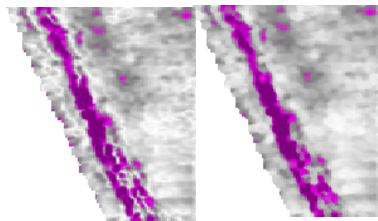
median filter: The median filter smooths smallscale heterogeneities but preserves edges. The filter size must be entered. 3*3 disk/li. and 3*3 disk includes 5 points, 3*3 square 9 points and 5*5 disk 13 points.

dilation/erosion filter. Dilation and erosion are morphology-based operations. Dilation, in general, causes objects to dilate or grow in size but also a hole fitting will be observed. Erosion causes objects to shrink. The combination of both has a similar effect like the median filter (see picture on the right, left panel original, right panel effect of dilation/erosion filter).

The filter size must be entered. 3*3 disk/li. and 3*3 disk includes 5 points, 3*3 square 9 points and 5*5 disk 13 points. The filter effect increases with the filtersize (3*3 disk/lin. has the smallest effect). The filter is mainly useful for the display of slices.

sharpen edges: if activated an algorithm will be applied for the plotting which sharpens the edges of the different elements within each image. The sharpen strenghts soft and strong can be entered.

square: if activated the data are squared when loading. The sign will be preserved and the resulting amplitudes will be normalized to the original maximum value. With the option **use mean** activated the normalization will be done based on the mean value. Using the option square allows a higher contrast. High amplitude objects will be emphasized in comparison to small amplitude reflections/diffractions.



There exist one possibility s to **output** the actual cuts:

GeoTiff export: allows to export the current cut to a tiff file together with a tfw world file. The x- and y-coordinates within the world file should be ignored either only one cut has been plotted. The format of the world file is:

pixel X size

rotation about the Y axis (usually 0.0)

rotation about the X axis (usually 0.0)

negative pixel Y size

X coordinate of upper left pixel center

Y coordinate of upper left pixel center

The tiff file size corresponds to the actual window size and the following plotoptions are automatically used:

no axis (plotoption showaxis automatically deactivated)

max. y-value on top (plotoption flip y-axis automatically activated)

A clipboard scale factor is queried. A factor larger than 1 allows you to increase the size of the bitmap to be transferred and therefore to increase the resolution.

The filename is automatically determined from the 3D-datafilename and the last current cut shown within the window. Example:

3D-datafilename: file01_3D.00t

last current cut: sclice: 0.58

resulting tiff-filename: file01_3Dt_0_58.tif

resulting word file: file01_3Dt_0_58.tfw

shift-3d-cuts: new option which allows to shift the cuts against each other. The option might be useful if the cuts exhibit a special zig-zag pattern (shift mode set to shift every 2. cut) due to e.g. a meandering data acquisition (see picture on the right) or an overall shift of special xy-ranges (shift mode set to shift every cut). The zig-zag pattern can only be removed if no interpolation has been used when defining the 3D-datafile.

It is possible to define up to 100 different rectangular xy-ranges (parameter number) whereby it is only possible to restrict the range perpendicular to the direction of the acquired profiles. The shift is only possible in the direction of the acquired profiles (not perpendicular to them). The shift will be displayed interactively as soon as any change of the parameters has been done. For that purpose the data should be displayed using the slices. It is possible to save the new 3D-datafile using the option analyse/shift-3D-cuts.

Input parameters:

number: number of the xy-range

Start x-cut or start y-cut: first cut number for the actual xy-range


end x-cut or y-cut: last cut number for the actual xy-range

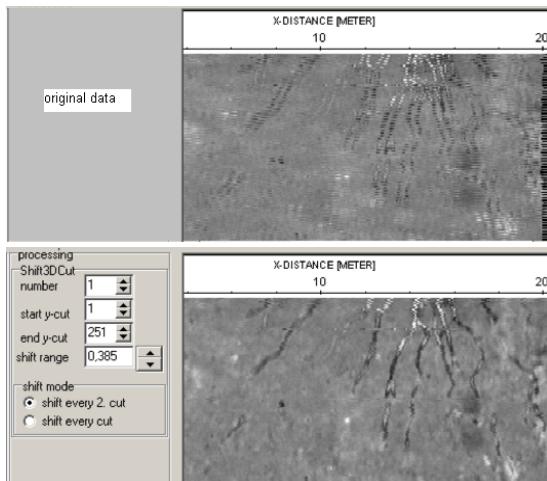
Shift range: input of the shift range using the given distance dimension.

shit mode: enter whether every 2. cut or every cut shall be shifted starting at start cut respectively.

The actual shift parameters can be saved on a file

shift3DCuts.fil under the project directory using the option . Those

parameters can also be reloaded using the option .



Using the option analyse/Shift 3D-cuts a new 3D-datafile containing the shifted cuts can be saved. For that purpose a processing label must be entered. The option is equivalent to the processing option Shift-3DFile under processing/trace interpolation within the 2D-dataanalysis which however uses the complete xy-range for the entered shift parameters.

Normalize 3d-cuts: allows an energy normalization for up to 10 different rectangular xy-ranges. The option might be reasonably employed if e.g. the timeslices show distinct stripe patterns which indicate different coupling conditions (see picture on the right). The energy normalization will be displayed interactively as soon as any change of the parameters has been done. For that purpose the data should be displayed using the slices.

Input paramters:

number: enter the number (max. 10) of the xy-range to be normalized


The xy-range is defined within the editfields **min.** and **max.**.

The z-range has been set to the max. range by default. It can be decreased for the normalization based on the constant normalizing factor.

Set to max: set the x/y cuts to the max. values for the actual 3D-profile

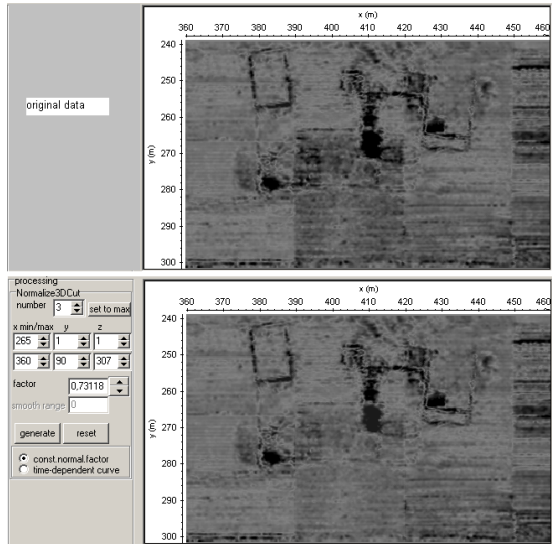
constant normalizing factor: one single scaling **factor** is used for all values lying within the chosen xy-range. Activating the option **generate** the scaling factor will be calculated from the quotient of the overall mean energy value of the 3D-datafile and the total energy value of the chosen xy-range. The mean energy values will be determined within the given timewindow (options min/max. z cuts). The factor can also be be entered (changed) manually.

time-dependent normalizing curve: a timedependent scaling curve will be calculated from the overall mean envelope curve of the complete 3D-file and the mean envelope curve of the chosen xy-range using the option **generate**. The envelope curves will be determined over the total timerange and summed up over all traces. Then this mean envelope (energy) curve will be smoothed over the given timerange (option **smooth range**). The option changes the energy distribution with respect to the timeaxis. For example: if a high reflection energy area exists within a special timerange within the chosen xy-range the option will lower the energy within this timerange for this range.

The actual normalize parameters can be saved on a file `normalize3DCuts.fil` under the project directory using the option .

reloaded using the option .

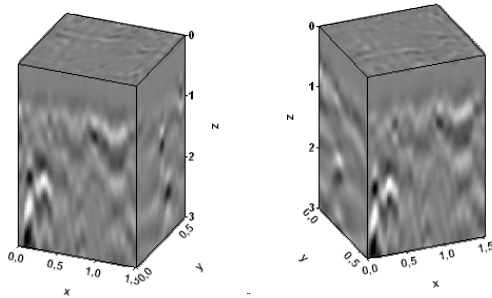
It is possible to save the new 3D-datafile containing the actual `normalize3DCuts` chnges using the option `analyse/shift-3D-cuts`.



3.4 3D-cube display

Using the option “3D-cube” allows the three-dimensional plotting of the currently loaded 3D-file within predefinable spatial limits (options **xmin**,...,**zmax**) and an arbitrarily definable observation point (**x°**, **y°** and **z°**). Thus you have the possibility to look at the cube from different angles, e.g. from the front side, the back side or from the top. The rotation is done interactively using the scroll bars.

To be considered: The directions of the x-, y- and z-axis do not follow the right hand rule because the z-axis points from top to bottom in positive direction. With the option **flip z-axis** activated the direction of the z-axis may be flipped. Then the z-axis points from bottom to top in positive direction. This also changes the orientation of the x- and y-axis. If you rotate the cube by 180° in x-direction the z-axis has again the same direction (from top to bottom in positive direction) but the x- and y-axis are exchanged (see picture on the right). If the option **flip z-axis** will be changed the rotation angles will be set on default values.



All three axis are automatically labelled. The scaling may be changed within the plotoptions using manual scaling and entering a new label increment for the different axis.

The option **black background** allows to choose a black background for the 3D-cube display.

By default the size of the cube depends on the number of points in the 3 different directions x, y and z (the z-direction is normally the time-direction). The size of the axis may be changed individually (options **scaleX**, **scaleY** and **scaleZ** - linear interpolation is done) and jointly (option **magnification** - no interpolation is done). Increasing the magnification does not coincide with an increase of the needed computer-time whereas increasing the **scaleX**, **scaleY** or **scaleZ** values leads to a significant increase of the computer-time.



These options allow the saving of the current 3D-cube display parameters(angles and scaling values and the magnification) and to read them from the disk (filename **cube3Dpar.txt** under the actual project directory).

When changing any of the parameters mentioned above the cube frame is always plotted in order to have an immediate control on the current settings.

You may select if only the front or back planes of the data cube are displayed (options **front** and **back**) or the full 3D-data volume (option **full**). In addition you

only may select single cuts (option **single**) and scroll (option **scroll**) through the cube in one distinct direction.

It is possible to **restrict** the datavalue range to be considered. Within the **value range** radiobox you may select if all data points (parameter **all** activated) or if only a special datavalue range (parameter **inside** or **outside** or **picks** activated) is considered. The datavalue range is defined within the editfields **min.** and **max.**. With the parameter **inside** activated all data values lying inbetween the specified min. and max. value are considered. With the parameter **outside** activated all data values lying outside of the specified min. and max. value are considered. With **picks** activated only these datapoints of the actual 3D-file will be plotted for which picks have been loaded (see also chap. 3.8.1). The option **fill traces** allows you to avoid gaps because of missing pick traces. Enter here a number of traces to be filled around the found pick trace, for example a value of 2 or 3 if small gaps are present.

The parameter **fill scale (%)** allows you to control the filling of the cuts. The option is not available for the full plot mode. By default the value is set to 100 %. The value range is restricted between 100 and 200 %.

It might happen that certain parts of a cut may not fully be covered by the used interpolation scheme. This results in white points normally arranged in a distinct pattern. In this case you may increase the fill scale parameter in order to avoid this problem. To be considered: Increasing the fill scale parameter results in a decrease of the resolution.

The option **hide** controls if not visible parts of profiles are covered (option activated) or not (option deactivated). With the option deactivated the max. amplitude value will be displayed.

The option **update** controls if the cube display is always updated when changing a parameter like roatation degree or scaling.

The option **plot** enables the plotting of the 3D-data volume based on the current settings.

With the option **front** activated only the front planes of the data cube are displayed when pressing the plot option.

In addition you may activate the **cube cutting** using the option **active**. The cornerpoint which serves as the starting point for a cutting out of the cube will be automatically determined dependent from the actual rotation of the cube if the option **keep corner point** is deactivated. With keep corner point active the actual corner point will be kept. It is recommended to deactivate this option at the beginning. The corner point is shown as a red circle. When displaying the data It is only shown if the option show border is active.

The size of this cube sector in the 3 directions (options x, y and z) can be changed individually or jointly (option synchronize activated). With the option **synchronize** activated the size in the other 2 directions are automatically changed when changing the size of any direction. The option **set to 0** resets the size to 0 and **set to max.** sets the size to the individual max. values. You can also use the cube cutting in order to scroll through the cube in combination with the display of the visible front or background planes. Example: use the option set to

max. and deactivate synchronize. Then you may change the z-size in order to scroll in z-direction.

With the option **back** activated only the back planes of the data cube are displayed when pressing the plot option.

In addition you may activate the **cube cutting** using the option **active**. The cornerpoint which serves as the starting point for a cutting out of the cube will be automatically determined dependent from the actual rotation of the cube if the option **keep corner point** is deactivated. With keep corner point active the actual corner point will be kept. It is recommended to deactivate this option at the beginning. The corner point is shown as a red circle. When displaying the data it is only shown if the option show border is active.

The size of this cube sector in the 3 directions (options x, y and z) can be changed individually or jointly (option synchronize activated). With the option **synchronize** activated the size in the other 2 directions are automatically changed when changing the size of any direction. The option **set to 0** resets the size to 0 and **set to max.** sets the size to the individual max. values. You can also use the cube cutting in order to scroll through the cube in combination with the display of the visible front or background planes. Example: use the option set to max. and deactivate synchronize. Then you may change the z-size in order to scroll in z-direction.

With the option **view contour** activated the contours of the cut planes are displayed in addition.

With the option **single** activated it is possible to plot any combination of x-,y- or z-cuts. You may enter single numbers separated by commas, blocks with dash or combination of the two (e.g. 5,6,7,10-20). The option step is used when the cuts are defined using numbers separated by a dash (e.g. 10-20, step rate: 2 - the following cuts are plotted: 10, 12, 14, 16, 18, 20).

With the option **scroll** activated it is possible to continuously move through the 3D-cube either in x-, y- or z-direction using the track bar. The **scroll** rate is freely selectable.

The control panel below the track bar allows an automatic scrolling through the 3D-cube:



scroll to smaller cuts



stop scrolling

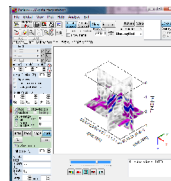


scroll to higher cuts

Use the options **X-Cuts**, **Y-Cuts** and **Slices** in the upper control panel to determine which 2D-cut of the 3D-file you want to scroll. With X-Cuts activated all existing cuts perpendicular to the x-direction may be chosen. With Y-Cuts activated all existing 2D-cuts perpendicular to the y-direction may be chosen. With Slices activated all existing 2D-cuts along the time-(z-)direction may be chosen.

It is possible to plot a "background" in addition. For that purpose you may define any x-, y- or z-cuts. You may enter single numbers separated by commas, blocks with dash or combination of the two (e.g. 5,6,7,10-20) - see also option single.

With the option **switch mode** activated the last chosen cut will



be kept if switched to another cut set (example switch from X-Cuts to Y-Cuts) . The option allows an easy detailed 3D-check with crossing cuts. With the option **view contour** activated the contours of the individuals slices are displayed in addition.

With the option **full** activated all the data of the 3D-cube are displayed. With the option **show processing box** deactivated the processing status window indicating the status of the rendering will not be shown. With the option **use envelope data** activated the envelope data are used for the full display in the case of "envelope timeslices only" used within the Ref3DProcessingForm. If deactivated the "normal" data are used. Within the **select interpol.direction** box you may define the interpolation direction for the 3D-full cube display. There are two possibilities to plot the whole data:

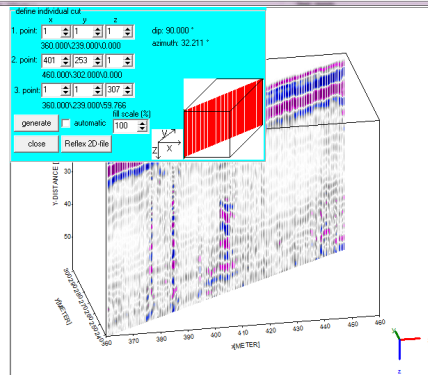
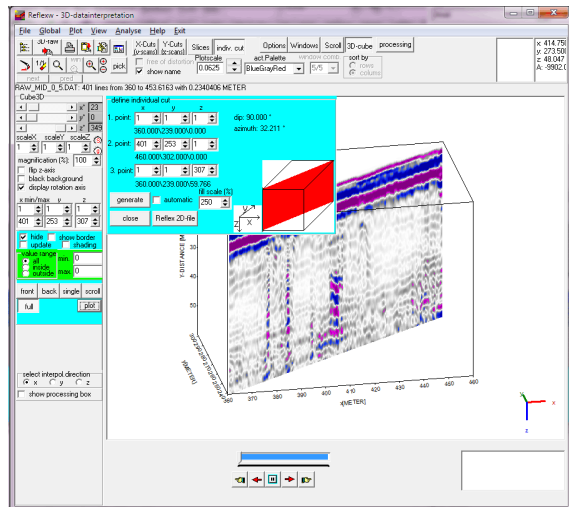
1. With the option **hide** activated and a **threshold** chosen, the data with having firstly an amplitude higher than the chosen threshold and secondly the smallest distance to the observation point will be plotted: By this means a possibility is given to "look through" certain parts of the 3D-data volume, whose amplitude values are smaller than the chosen **threshold** value. Using the option **shading** in addition a shading algorithm is used to give certain objects a contour. Shading is based on the spatial derivatives of the distances from the observation point. Data points with a small change rate are lightened whereas data points with a high rate are darkened. The min. and max. change rates are taken from the data. In many cases some very high change rates will occur. Therefore filtering of the change rates is necessary (box **use filter for shading**). You may choose between **no** (no filtering), **sqrt** (square root), **ln** (natural logarithm) or **median** (the max. value is restricted to twice the median), which is the standard filter option. To display the data, it is possible to use the **current color palette** together with a contrast value or a **gray, blue or red shade palette** from bright to dark. If you are using the gray shade palette the data points with the smallest distance change rates are white and the data points with the highest change rates are black. If you are using the current color palette you must enter the **contrast** value. The color associated with the current data point is brightened or darkened according to the relative change rate and the entered contrast. The max. contrast value is 255.
2. With the option **hide** deactivated the maximum amplitude will be plotted by taking into consideration only the chosen **threshold** and not the distances to the observation point. Therefore, no shading is possible in this case. For instance, this option can be used to "cut open" line or point diffractors, respectively, in migrated data. Assuming that the energy increases up to the middle of the diffractor and then decreases again, a point diffractor, e.g., will be displayed in plane view (x-y-plane, e.g.) as concentric circles with increasing amplitudes up to the middle.

The other parameters like envelope, absolute values,..., are described under Scroll 3D-File (chap. 3.2).

indiv. cut:

the option allows the definition and the display of an individual cut. The definition is done by defining 3 points of the plane. The definition is given in samples in x-, y- and z-direction. The corresponding coordinates are automatically updated below as well as the dip and the azimuth. The azimuth is calculated from the intersection of the plane with the xy-plane and the x-direction. Activating the option **generate** shows the actual cut plane within the sketch cube. If **automatic** is active the sketch cube will be updated automatically when changing any cut point. The individual cut will be plotted into the 3D-cube using the option **plot** with the 3D-cube plotoption full activated. The actual parameters will be saved after having closed the indiv.cut menu (option close). Normally the gridded datapoints do not exactly fall into the chosen plane. Therefore adjacent datapoints next to the plane must be taken into account in addition. The size of region is defined by the parameter **fill scale (%)**. If set to 100 datapoints which are next to the chosen plane by half of the gridincrement are used. Increasing this value takes more datapoints into account. If the cut is not fully filled (see picture on the right) increase this value, until the filling is complete. The necessary fill scale value also depends on the actual display view.

The option **Reflex2D-file** generates a Reflex 2D_file of the actual cut. Again the fill scale (%) must be adapted. In many cases a smaller value is sufficient (decrease the value if the 2D-file contains too many traces). The filename is automatically determined from the 3D-datafilename and the current dip and azimuth and will be stored under the path rohdata under the actual projectdirectory. Example:
 3D-datafilename: raw_mid_0_5.DAT
 dip: 90° and azimuth: 32.211°
 resulting Reflex 2D-filename: raw_mid_0_5_90_000_32_211.DAT
 If the wanted cuts have the same dip and azimuth you must rename the 2D-files manually.



3.5 Generate 3D-file from 2D-lines

The option allows to construct a 3D-datafile either from equidistant parallel 2D-lines without interpolation or from 2D-lines freely orientated within one acquisition plane using a weighted interpolation.

The filename of the resulting 3D-file must be entered under **3D-filename**. The 3D-datafile is stored under the path ROHDATA under the projectdirectory.

Generate 3D file from 2D-lines

3D filename: test3d

type of interpolation: use interpolation scheme for freely distributed 2D-lines

☐ add xy-scans by choosing the max. abs. value

☒ only those files in time-direction

3D coordinates

XStart: 461771.3625

XEnd: 461802.2217

YStart: 5427196.346

YEnd: 5427268.079

XRasterincement: 0.1

YRasterincement: 0.1

Zinterpolation: 1

Yinterpolation: 1

interp weight: square weight

data scaling: 1

interpolate/forcing

time end: 50

timeincrement scale: 1

scaling: recover coordinates

3D files choice

filter: procdata

load 2D-files: DAT_0001.00T, DAT_0002.00T, DAT_0003.00T, DAT_0004.00T, DAT_0005.00T

min/max xy coord: DAT_0004.00T

2D-lines statistics: DAT_0005.00T

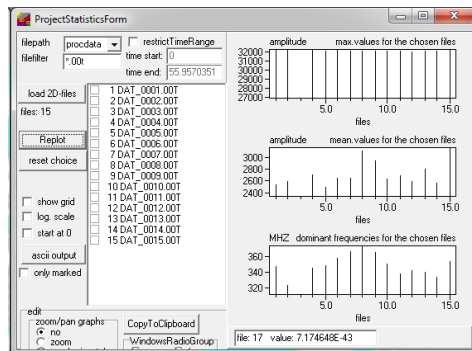
☒ update headreader coordinates

Start show lines Cancel

The option **load 2D-files** allows to specify the wanted 2D-files for the generation of the 3D-file. You must choose the original data path (either rawdata or procdata) and afterwards you must select the wanted 2D-files.

The option **Show lines** allows to displays the xy-geometry of 2D-lines currently loaded.

The option **2D-lines statistics** opens a new window where the currently loaded 2D-lines may be analysed in different ways. One useful application of the option might be a check of the energy homogeneity between the different 2D-lines, e.g. if an antenna array has been used for acquiring parallel 2D-lines. It also may serve as an overall overview about the amplitude distribution, e.g. before generating a 3D-datafile for which the datavalues should not exceed a value of 32000 (16 bit integer limitation).



The parameter **type of interpolation** defines which type of interpolation is used.

After having chosen **equidistant parallel 2D-lines without interpolation** the 3D-file is generated from equidistant parallel 2D-lines. Precondition is the same traceincrement and timeincrement for all 2D-lines.

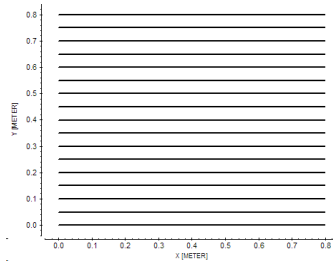
The **line-increment** specifies the increment between successive parallel lines in the given distance dimension of the 2D-lines.

The radio buttons **sort by filenames** and **sort by coordinates** specify how the 2D-files are sorted for the generation of the 3D-file.

With **sort by filenames** activated the 2D-files are sorted by files with ascending order. The chosen lines are alphabetically resorted (sorted with ascending order - see also Import ConversionMode). Problems may occur if the files don't have the same length (e.g. file1, file2,..., file11). In this case the chosen files are resorted in the following manner: file1, file11, file2, Please take care that all original filenames have the same length - for example: file01, file02, ..., file11. The coordinates are redefined based on the entered line-increment and the profile constant start-coordinate of the first chosen line (after alphabetical resorting). The start coordinate in profile direction of each 2D-line is taken into account. For this purpose the minimum start coordinate of all 2D-lines is calculated and zero traces are inserted for those 2D-lines which start coordinate is larger than this minimum coordinate. All other previously entered coordinates of the 2D-lines are of no importance.

With **sort by coordinates** activated the 2D-files are sorted by the coordinates in profile constant direction with ascending order.

A 3D-data file consisting of equidistant parallel lines (sorted by filenames) may also be constructed directly during the data import (- see also Import ConversionMode - option combine lines/shots).



After having chosen **use interpolation scheme for freely orientated 2D-lines** you may construct a 3D-file from freely orientated 2D-lines analogous to the generation of single timeslices (see also Generate single timeslices).

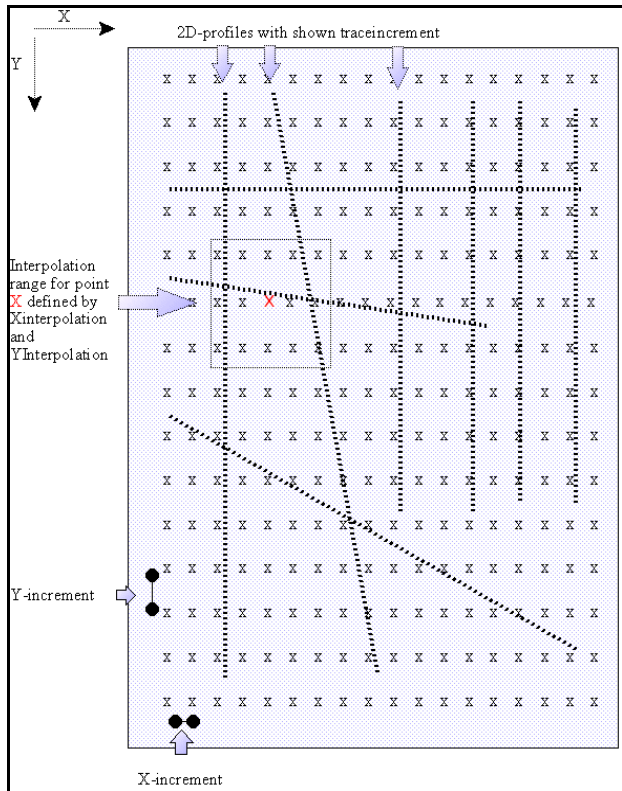
The parameters within the **3D-coordinates** group box allow the settings of the raster.

The parameters **XStart**, **XEnd**, **YStart** and **YEnd** specify the range coordinates within the dataacquisition plane for the computation of the 3D-data cube. The dimension complies with the distance dimension of the individual 2D-profiles. This has to be the same for all profiles, of course. To be considered: The program uses 32 bit floating point representation for these values. Therefore if very high values are entered (e.g. values like 40000000) a warning message appears that the data might be smoothed.

The next two parameters **XRasterincrement** and **YRasterincrement** specify the grid interval of the two coordinate axes spanning the plane in the given distance dimension. This grid interval determines the discretization of the 3D-data cube and thus also the size of the file. In both coordinate directions the max. number of points can be 2048. A

useful value for both **XRasterincrement** and **YRasterincrement** is the trace increment used for the individual 2D-profiles but it might be that a greater value must be used (e.g. if the resulting 3D data-volume becomes too big).

The next two parameters **XInterpolation** and **YInterpolation** define the inter-polation area in the given distance dimension. Due to the specification of the interpolation range in the two directions for each discrete point of the timeslice a rectangle is opened. All points of the individual 2D-profiles, falling in this rectangle, are considered and the distance to the discrete point is determined. The



Sketch of a data acquisition and a subsequent 3D-block generation with extremely distributed 2D-lines and very large interpolation range

individual amplitude values from the desired time range (see below) are weighted according to the distance, summed and assigned to the discrete point. To guarantee a complete filling of the 3D-data cube, the interpolation in x- or y-direction has to comply with the greatest occurring distance between two data points in the corresponding direction. The parameter **interpol.weight** allows to specify the type of the interpolation weight. Using **linear weight** means that the values are weighted according to the distance, using **square weight** means that the values are weighted according to the square of the distance. The option **data scaling** allows to enter a scaling factor with which the resulting 3D-datafile values will be multiplied. This might be necessary because during the 3D-datafile generation the data will be stored as 16 bit integer values and consequently may fall out of the binary data representation if the original data exhibit small (e.g. smaller than 1) or very large values (greater than 32000). A corresponding data scaling value (greater 1 for small original data values and smaller 1 for great original data values) can be used in order to rescale the data in order to be able to be represented as 16 bit integer.

The option **min/max. xy coord.** allows to extract the Xstart, Xend, Ystart and Yend coordinates from the file- or traceheaders of the loaded 2D-files depending on the setting of the sorting option. The min./max. coordinates exactly define the borders of the acquisition area. Therefore it might be useful to extend these values a little bit.

The parameters within the **timerange/sorting** group box allow the settings of the timerange and the sorting.

The parameter **time end** specifies the timerange for the 3D-data cube. The time always starts at 0. Therefore all 2D-lines also should start at time 0 (time begin must be 0, if not please use the processing option move starttime for the 2D-lines before). The parameter **timeincrement scale** allows to use a different timestep for the 3D-data. The timeincrement scale is a multiplication factor for the original time step within the 2D-data. It should be set to values greater than 1 if the step within the 2D-data is quite small and the sample number quite high.

The parameter **sorting** determines the sorting of the profiles entering the computation of the time slices. Four different sortings are possible:

fileheader coordinates - the program takes the current trace coordinate from the profile headers (x, y, z begin - and end coordinates, profile direction and profile constant). This sorting is recommended if zero-offset profiles are used for time slice computations (normally the case for GPR-measurements).

Receiver coordinates - the current coordinate is read from the receiver coordinates of the individual traces (see also trace header Edit).

midpoint coordinates - the current coordinate is calculated from the shot and receiver coordinates of the trace header (midpoint between shot and geophone - see also trace header Edit).

CMP coordinates- the current coordinate is taken from the CMP-coordinates of the trace header (see also trace header Edit).

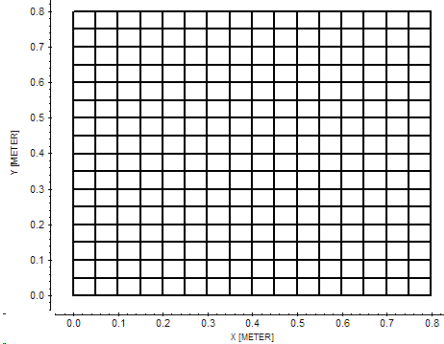
The traceheaders of the resulting 3D-file contain the new coordinates except the option **update traceheader coordinates** has been deactivated when using the type of interpolation "equidistant parallel 2D-lines without interpolation". In this case the old traceheader coordinates will be kept.

After having chosen **use independent interpolation scheme for pure x- and y-2D-lines** you may construct a 3D-file from 2D-lines which are purely or nearly purely distributed along the x- and y-axis. Using this option allows a much better generation of a complete 3D-file from crossing lines.

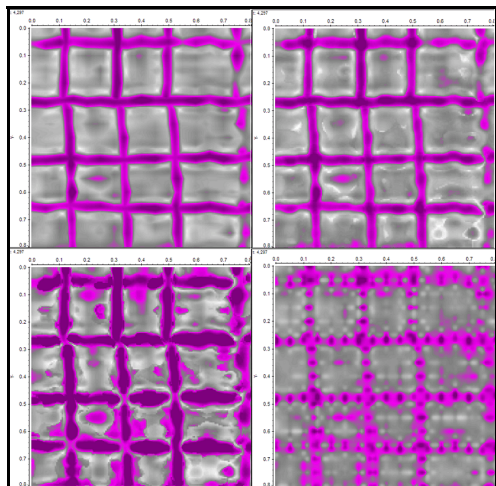
The input parameters are the same as for the type **use interpolation scheme for freely orientated 2D-lines** except the option **apply mean filter in time-direction** which allows a smoothing in time direction. The smoothing value is given in samples (+- samples symmetric around actual point). This smoothing prevents the destroy of the smoothed signal form when adding two traces at the crossing points.

The following example shows the comparison of different 3D-datasets:

- upper left: overlay of two 3D-datasets which have been created only from x- and y-scans
- upper right: 3D-data created from both x- and y-scans using the type “use independent interpolation scheme for pure x- and y-2D-lines” with activated option apply mean filter with a size of 4 samples
- lower left: 3D-data created from both x- and y-scans using the type “use independent interpolation scheme for pure x- and y-2D-lines” with deactivated option apply mean filter
- lower right: 3D-data created from both x- and y-scans using the type “use interpolation scheme for freely distributed 2D-lines”. For the input x- and y-scans the envelope data had been used.



The comparison shows that the 3D-file created using the type “use independent interpolation scheme for pure x- and y-2D-lines” with activated option apply mean filter is comparable to the overlaid independent 3D-datasets which serves as the reference (upper pictures). A smoothing in time direction is necessary (lower left panel without smoothing). The overall interpolation not taken into account the original profile direction (lower right panel) gives a much worse result even if the original data are available as envelope data.



3.6 Generate single timeslices

The computation of timeslices (C-scans) allows the representation of equitemporal amplitude values. The precondition is a full coverage of the investigation area. The 2D-profiles entering the calculation have to originate all from one plane and must be static corrected. The profiles can have arbitrary orientations. Gaps between the profiles are filled by a corresponding interpolation.

To be considered: The amplitudes are rescaled during the calculation of the timeslices depending on the current interpolation and summing parameters. If you wish to compare different timeslices with respect to the amplitudes, all timeslices have to be calculated using the same parameters (raster increment, interpolation points and time range, which is used for the summing).

The parameters within the **coordinates** group box allow the settings of the timeslice raster.

The parameters **XStart**, **XEnd**, **YStart** and **YEnd** specify the range coordinates within the dataacquisition plane for the computation of the timeslice. The dimension complies with the distance dimension of the individual 2D-profiles. This has to be the same for all profiles, of course. To be considered: The program uses 32 bit floating point representation for these values. Therefore if very high values are entered (e.g. values like 40000000) a warning message appears that the data might be smoothed.

The next two parameters **XRasterincrement** and **YRasterincrement** specify the grid interval of the two coordinate axes spanning the plane. This grid interval determines the discretization of the time slice and thus also the size of the file. In the second coordinate direction (in this example the y-coordinate) the number of points can be 16384 maximum, in the first coordinate direction 64000. A useful value for both XRaster and YRaster is the trace increment used for the individual 2D-profiles.

The next two parameters **XInterpolation** and **YInterpolation** define the interpolation area in the given distance dimension. Due to the specification of the interpolation range in the two directions for each discrete point of the timeslice a rectangle is opened. All points of the individual 2D-profiles, falling in this rectangle, are considered and the distance to the discrete point is determined. The individual amplitude values from the desired time range (see below) are weighted according to the distance, summed and assigned to the discrete point. To guarantee a complete filling, the interpolation in x- or y-direction has to comply with the greatest occurring distance between two data points in the corresponding direction. The parameter **interpol.weight** allows to specify the type of the interpolation weight. Using **linear weight** means that the values are weighted according to the distance, using **square weight** means that the values are weighted according to the square of the distance.

With the option **interpolate for pure x-and y-lines** activated the choosen 2D-lines are assumed to be purely or nearly purely distributed along the x- and y-axis. In this case the interpolation will be done within 2 steps. The first interpolation is only done in x-direction for the profiles orientated into the y-direction. The second interpolation is only done in y-direction for the profiles orientated into the x-direction. Afterwards both datasets will be added.

The parameters within the **timerange/sorting** group box allow the settings of the timerange and the sorting.

The parameters **time begin**, **time end** and **number of slices** specify the time range(s) for which the time slice(s) is (are) computed. With the setting of number of slices the number of time slices to be computed with the above spatial parameters is determined. With the number of time slices to be computed the temporal range specification (time begin to time end) is divided into a corresponding number of equidistant intervals for which the time slice calculation is performed. The first computation begins at time begin and ends at time begin+(time end - time begin) divided by the number of time slices. The computation of the next time slice begins at the end of the first and so on. The time dimension has to be identical for all profiles.

The parameter **summing** determines how the time samples within the predefined time range are summed. You have the possibility of a summation of absolute amplitudes or of true amplitudes or no summing or of choosing the max. absolute amplitude value within the timerange. The summing of absolute amplitudes or choosing the max. absolute amplitude is recommended, if the summation range is larger than half a wave length and/or the arrivals cannot be static corrected exactly. No summing means that for each time sample within the given time range an individual timeslice will be created. The parameter number of slices has no meaning in this case.

The parameter **sorting** determines the sorting of the profiles entering the computation of the time slices. Four different sortings are possible: fileheader coordinates - the program takes the current trace coordinate from the profile headers (x, y, z begin - and end coordinates, profile direction and profile constant). This sorting is recommended if zero-offset profiles are used for time slice computations (normally the case for GPR-measurements).

Receiver coordinates - the current coordinate is read from the receiver coordinates of the individual traces (see also [trace header Edit](#)).

midpoint coordinates - the current coordinate is calculated from the shot and receiver coordinates of the trace header (midpoint between shot and geophone - see also [trace header Edit](#)).

CMP coordinates- the current coordinate is taken from the CMP-coordinates of the trace header (see also [trace header Edit](#)).

With the option **use envelope** activated the envelope will be used for the generation of the timeslices.

The option **load 2D-files** allows to specify the wanted 2D-files for the calculation of the timeslices. The maximum number of 2D-files is restricted by the system (about 1000 files). This number is not exact and therefore no warning message appears.

The option **load from ASCII** allows to specify the wanted 2D-datafiles within a separate ASCII-file. Each line of the ASCII-file contains the name of the 2D-datafile without the filepath. The complete filename of each 2D-datafile is given by the current project directory, the chosen filepath (procdata or rawdata) and the filename stored within the ASCII-file. This option overcomes the problem with specifying more than about 1000 2D-files using the option load 2D-files.

The option **min/max. xy coord.** allows to extract the Xstart, Xend, Ystart and Yend coordinates from the file- or traceheaders of the loaded 2D-files depending on the setting of the sorting option. The min./max. coordinates exactly define the

borders of the acquisition area. Therefore it might be useful to extend these values a little bit.

The parameter **filename specif.** is used for the naming of the computed time slice.

manual /automatic1: the naming is based both on the manually entered **filename** and an automatically determined part. For the manual name up to 30 characters are permitted. If you use less than 2 characters, the name is filled with '_.'. The following 6 digits are taken from the time range beginning of each time slice. Based on the max. travelttime of the original data a multiplication factor 1,10,100 or 1000 is calculated in order to fill out completely the four digits. The greatest power of ten of the current time coordinate is multiplied by this factor. Thereby it is possible to take into account timeslices with starting times differing from each other only slightly (e.g. 4.5 and 4.7 nsec).

manual /automatic2: the naming is based both on the manually entered **filename** and an automatically determined part. For the manual name up to 28 characters are permitted. If you use less than 2 characters, the name is filled with '_.'. The following 6 digits are taken from the time range beginning of each time slice. Based on the max. travelttime of the original data a multiplication factor 1,10,100 or 1000 is calculated in order to fill out completely the four digits. The greatest power of ten of the current time coordinate is multiplied by this factor. Thereby it is possible to take into account timeslices with starting times differing from each other only slightly (e.g. 4.5 and 4.7 nsec). The 7th and 8th digit of the name are according to the **file number**, as explained under automatic name.

automatic name: the automatic file naming is used. The temporal beginning of each time slice is used for the 2.-6. digit analogous to the manual naming. The 1. character is a 'T'. The 7th and 8th digits of the name are determined by the so called **file number**.

The timeslice is saved within the directory Procdata and automatically receives the processing label 0.

Example for automatic naming:

settings: time begin: 100 msec, max. time for calculation of timeslices: 500, number: 0 - time slice receives the name TP100000.00T (data file) and TP100000.00R (profile header file) and is saved in the directory Procdata.

Example for manual naming:

settings: time begin: 5 nsec, max.time: 50 nsec, FileName: Test - time slice receives the name Test0500.00T (data file) and Test0500.00R (profile header file) and is stored in the directory Procdata.



Enter this option, if you want to change the plot parameters and to save them in the fileheader of each current timeslice.

The following options control the generation of an optional additional **geotiff file**:
create Geotiff files: generates a tiff file together with a tfw world file containing the coordinates.

no timeslices .files: no timeslice file will be generated. This is only useful if create Geotiff files has been activated

full screen for geotiff: the normal size of the tiff file corresponds to the size of the picture above. If full screen for geotiff has been activated the full size of the

GenerateTimeslices window will be used for the tiff file. This allows a higher resolution.

plotscale: enter the plotscale for the display of the timeslice. If a Geotiff file will be generated the plotscale must be correctly entered here because the bitmap cannot be changed afterwards.

save: saves the current timeslice parameters on an ASCII-file with the extension tsl. If create Geotiff files has been activated a geotiff file (extension tif) together with a world file (extension tfw) will be saved under the directory ASCII in addition.

load: loads previously saved timeslice parameters from an ASCII-file with the extension tsl

create: start the calculation of the timeslices.

batch create: batch start for the calculation of the timeslices. Precondition is that different timeslice parameter ASCII-files (see option save) have been created before. In order to discriminate the individual filenames of the timeslices it is recommended to use the filename specif. manual/automatic. Of course the manual filenames have to be defined within the ASCII-timeslices parameter file before. After having activated the option the timeslice parameters ASCII-files are queried (multiple file choice possible). All timeslices are calculated for the current chosen set of 2D-files.

showlines: show the 2D-lines currently loaded.

3.6.1 Timeslice calculation example

The measured area has a size of 50(x)*40(y) meter. 20 parallel 2D-lines with a length of 50 meter in x-direction and an offset in y-direction from each other of 0,5 meter have been acquired. The recorded time length is 50 nsec. The trace increment within each 2D-line is 0,05 meter.

Timeslices are to be calculated based on the following parameters:

x-start:	0
x-end:	50
y-start:	0
y-end:	40
XRasterincrement:	0.05
YRasterincrement:	0.05
XInterpolation:	0.05
YInterpolation:	0.50
time begin:	0
time end:	50
number of slices:	20
sorting:	fileheader coordinates
summing:	absolute amplitude
filename specif.:	automatic name
filename:	
filenumber:	0

20 different timeslices are calculated. Each timeslice is summed over a time range of 2,5 nsec. The number of interpolationpoints in y-direction is greater than in x-direction because of the big offset between each parallel profile of 0,5 meter. 0.5 meter is the minimum value in order to completely fill the measured area. It is also possible to enter greater values for the interpolation, because the interpolation is based on a weighted summation. The following values are also useful:

XInterpolation:	0.1
YInterpolation:	1.0

3.6.2 Timeslice calculation overview

The computation of timeslices (C-scans) allows the 2-dimensional representation of amplitude values at equal times. The precondition is a full coverage of the investigation area.

The computation of time slices is performed within the module 3D-datainterpretation using the option Generate single timeslices within the 3D-File MenuItem.

The time slice allows to recognize reflecting elements lying in one depth (time). The profiles entering the calculation all have to originate from one plane and a static correction has to be performed. Before the computation of the time slice the desired profiles have to be selected interactively from the profile location map.

The profiles can have arbitrary orientations. The plane can represent either the x-y, the x-z or the y-z-plane. In the following we restrict ourselves to the x-y-plane for the description. For the other planes the equivalent applies.

The geometrical location of the profiles is determined by the defined coordinates, i.e. by the start coordinate in profile direction, the trace increment, the start coordinate in the direction of the profile constant, and the end coordinate in the direction of the profile constant. Normally the x-axis or the y-axis should comply with the profile direction. "Oblique" profiles, i.e. profiles that do not lie along an axis, nevertheless can be introduced into the computation of the time slices. The program refers to the coordinate specifications, then.

For the computation of the time slices first the extension and the gridding of the region for the time slice in x- and y-direction have to be specified. In the first step the program determines for all selected profiles the amplitude values pertaining to the selected time range and assigns them to the corresponding grid point. If several values fall on one grid point (e.g. in the crossing point of single profiles), the mean value is calculated. If the preset grid interval is smaller than the distance between the profiles and/or the trace interval within the profiles, these gaps are filled by an appropriate interpolation. For this you have to specify the interpolation range in x- and y-direction. Thus, around each grid point within the plane a rectangle of the size x-points*y-points is put and all amplitude values within this rectangle are weighted according to their distance to the current grid point and then summed. The result is a more or less smoothed (according to the set rectangle) area-covering representation of equitemporal amplitude values. Normally, it is recommended not to compute the time slice only for one single time sample but to average over a certain time range (approx. 1/4 wave length), since because of velocity fluctuations in the upper layers the arrival times of a reflection from profile to profile are often submitted to certain fluctuations.

The time slices are stored in the same format as the "normal" sections, so that any data processing and visualization possibilities can be applied. If you view the time slice as a profile, the position of the profile is rotated against the "normal" zero-offset profiles. Per definition for each computed time slice for the profile direction the x-axis is set as well as for the profile constant the z-axis is set. "Time" corresponds to the y-axis. Of course, this only applies to zero-offset profiles in the x-z-plane. The other planes are treated analogously.

The time slices can be constructed either from parallel profiles or from profiles crossing each other, as it is usual, e.g. for GPR investigations, or from any traces located in the plane, which originate from special shot-receiver-geometries. For sorting (geometry) the data the program can either access the coordinate values

saved in the header or the shot and receiver coordinates stored in the trace header (if these exist - check over HeaderInfo in Data processing). For equidistant zero-offset-profiles one would normally use the header coordinates.

3.7 view fast data cube display

The option fast data cube display under view within the 3D-datainterpretation opens the fast 3D- cube window which allows you a 3D-view of the currently loaded 3D-file. It supports an interactive rotation of the 3D cube. The rotation can be achieved either by mouse interaction or by using scroll bars. The data can be viewed from any direction and can be zoomed.

It is possible to bring this window to the front by pressing the right mouse button within the data window of the 3D-datainterpretation.

For the fast data cube display first the data are rescaled and loaded into an internal buffer. Then the data are ready for the display and the rotation. The time needed for the rotation or the display depends on the size of the 3D-data volume and on the complexity of the display, i.e. the number of different colors. Therefore you have different possibilities to restrict the total number of datapoints to be displayed (e.g. amplitude threshold, definition of the polarity and defining the max.nr. of points).

All three axis of the 3D-cube have the same length. The z-direction always corresponds to the time-(y-)axis of the 3D-data.

The scroll bars at the bottom determine the angles of view. The current angles in x-, y- and z-direction are displayed on the right. In addition you may use the mouse for the **rotation**. In that case only the x- and y-angles can be changed, the z-angle is fixed to 90 °.

Some examples for the rotation around one axis:

rotation around z-axis: $Z=0^\circ$, $x=270^\circ$ and use the y-angle for the rotation.

rotation around x-axis: $Z=0^\circ$, $y=0^\circ$ and use the x-angle for the rotation.

rotation around y-axis: $Z=0^\circ$, $x=0^\circ$ and use the y-angle for the rotation.

The magnification is controlled by the option **magnification** (in %) .

First you have to **load** the data into the internal buffer.

The option **amp. threshold** defines an amplitude threshold for the data. Only those data having an amplitude larger than this threshold are stored within the internal buffer.

Within the **polarity box** you may define the polarity of the datapoints to be loaded into the internal buffer. This box is only enabled if both the option absolute amplitudes and envelope are deactivated within the 3D-datainterpretation.

The option **max. nr. of points** defines the maximum of data points stored within the internal buffer. If the number of datapoints exceeding the given amplitude threshold is greater than this value the program automatically determines a factor n for reducing the data. Only every n. datapoint will be stored within the internal buffer.

The number of datapoints currently loaded into the internal buffer is shown after having pressed the option load.

Within the **colors** box you may define if all data are displayed in blue or using the current color palette. To be considered: for performance reasons only 16 different colors from the current color palette are used.

The option **mark size** allows you to define the size of the display marks. Use a big size for a small datavolume and use a small size for a big datavolume.

The option **clear** empties the internal buffer. The rotation is much faster with an empty buffer.

The option **print** allows you to print out the current 3D-cube. The option **print scale** controls the size of the output.



copy 3D-cube to the clipboard

The option **close** closes the fast data cube display window.

3.8 3D picking

With the pick option activated arrivals of a 3D-datafiles can be picked analogous to the pick option within the 2D-dataanalysis (see also Pick Menuitem, chap. 1.12.2). Picking may be done within the individual 2D-cuts (scroll or window mode) or within the 3D-datacube display. You have the choice between manual picking and continuous pick.

Corrections can be performed either on zero crossings, on the extremum or some other types (options correct all or corr. act cut). The picks can be saved with an arbitrary name and reloaded any time (option save and load). The maximum number of picks is limited to the maximum number of scans per profile.

The option take over allows you to take over the picks of the previous cut for the current one. This allows you a fast picking of the individual cuts.

With the option use code activated you may pick all elements within one step (the pick code is used for discriminating the individual elements (reflectors)).

Both the travel time and the current amplitude of the phase are detected. In addition to the distance the xyz-shot- and receiver coordinates of each individual trace are stored within the pick record.


Within the global settings menu, chap. 1.2.1 you can choose between two different symbols for the picks (* and -). The size of the pick-symbol can be changed within the symbol font menu (see also Global Menuitem).

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

continuous pick: With this options activated you can set (option set activated) or change (option change activated) picks by moving the mouse cursor over the data set with continuously pressed left mouse button. The picks are set or removed or changed at those traces that are next to the current cursor position. The continuous picking must be done from left to right, this means with increasing distance coordinates. The option continuous pick therefore makes manual picking a lot easier for large data sets. If the cursor movement is too fast to pick every trace no interpolation between the founded pick position is done in contrast to the 2D-dataanalysis. The parameter delay under global settings controls the speed of the continuous picking.

show xy-proj.: interactive display of the xy-projections of the current picks. The option is useful for a direct control of the locations of the picks within the xy-plane. Picks belonging to the same code are connected if the option interpolation is activated.. For example the option might be used for the location control of picked pipes or rebars If deactivated no connection will be done.

3D-pick-cube: interactive fast 3D cube display of the current picks. The 3D-cube may be freely rotated either by mouse interactions or by using scroll bars. The picks can be viewed from any direction and can be zoomed. It is possible to bring this window to the front by pressing the right mouse button within the data window of the 3D-datainterpretation. The option **print** allows you to print out the current

3D-cube. The option **print scale** controls the size of the output.  copy 3D-cube to the clipboard. The magnification is controlled by the option **magnification** (in %).

pick code: input of a code (integer value) for the characterization of the following picks to be chosen. With the option **use code** activated this code may be used for discriminating the individual layers or structures. The current code of already existing picks is displayed when moving the mouse cursor (label act - code of the pick next to the mouse cursor).

use code:

if activated the pick code is used for discriminating the individual picks:

- only the picks with the current code may be changed or removed (the current code is automatically updated when moving over a pick if the option only act code is deactivated)
- different colors are used for the picks with different codes (by default the layershow colors are used, therefore the max. code is restricted to 99)
- different correction types can be used for the different codes. The option apply to all codes allows to apply the current correction type to all pick codes.
- options corr. act cut, correct all and auto corr. only act on the picks with the current code if the option **only act code** is active in addition (if deactivated all picks are used)
- options reset and reset act cut only act on the picks with the current code if the option **only act code** is active in addition (if deactivated all picks are used)
- options control all and control act cut only act on the picks with identical code if the option **only act code** is active in addition (if deactivated all picks are used)
- the colors as well as the correction type are hold within the RAM for each code (option show shows you the current settings)
- option view/Pick surfaces for 3D-cube: code is used in order to discriminate the individual layers

if deactivated the following holds true:

- all picks may be changed or removed
- the current pick color is used for all picks
- one correction type is used for all picks
- options control act and control all: only picks which belong to a special trace and which traveltimes are nearly identical are controlled and if necessary double picks are removed
- option view/Pick surfaces for 3D-cube: the layernumber stored with the pickfile is used for the discrimination of the individual layers

show: shows the current settings of the pick codes (color and correction type)

color: enter the wanted pick color

load: This option allows to load new picks from disc. Please note that the current picks are lost with the loading of new picks. After selecting the option you can choose a data set from the file list. In addition to the REFLEX pick format two ASCII-format are supported:

ASCII-2 columns: each line of the ASCII-file corresponds to one pick and consists of two columns containing the distance and the time in the corresponding dimensions. Depending on the actual settings of the cut-mode (x-, y-cut or slices)

the distance and the time value will be used for the corresponding actual x- and y-axis.

ASCII-3D-tomography: corresponds to the ASCII-3D tomography format described within the Pick save menu (chap. 1.12.2.1).

save: saves the current picks. After selecting the option the Pick save menu (chap. 1.12.2.1) will be opened.

reset: This option allows to reset all current picks. With the options use code and only act code activated only the picks with the current code are lost.

reset act cut: This option allows to reset the picks of the current cut. With the option use code and only act code activated only the picks with the current code are lost.

take over: this option allows the picks of the previous cut to be taken over for the current cut. As long as this option is activated the picks are automatically taken over for the next cut (use for example the single scroll bar). With the options use code and only act code activated the option only acts on the picks with the current code. With the option auto cor. activated the actual correction type(s) is (are) used. The option will be automatically deactivated when loading new picks.

only act code: if activated the options take over, correct all, corr. act cut, auto cor., control all and control act cut as well as the move up and down arrows only act on the picks with the current code.

correct all: activate this option to correct the current picks using the current correction type. With the options use code and only act code activated the option only acts on the picks with the current code.

corr. act cut: activate this option to correct the current picks of the current cut using the current correction type. With the options use code and only act code activated the option only acts on the picks with the current code.

The following correction types are available within the combo box above:

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

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

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

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

dip correct: the picks are corrected for the dip of the reflector (dip is in scroll direction). The dip is automatically calculated from the mean of the traveltimes differences of the reflector picks of the two previous cuts which are placed at the same distance position. If no picks with the same distance position exist for the two cuts the dip correction is set to 0.

auto cor.: if activated the picks are automatically corrected during the setting. The current correction type is used. With the options use code and only act code activated the option only acts on the picks with the current code.

apply to all codes: The option allows to apply the current correction type to all pick codes.

The two **arrows** at the right side allow to move up and down the picks of the current cut by one sample increment. With the options **use code** and **only act code** activated in addition only the picks with the current code are moved.

control all: this option allows you to remove double picks, this means several picks for one distance (one trace). With the options use code and only act code activated only double picks with identical codes are removed.

control act cut: this option allows you to remove double picks of the current cut. With the options use code and only act code activated only double picks with identical codes are removed.

interpolate to 3D:

The option allows to interpolate the actual picks based on the actually loaded 3D-file. With the option **restrict to pick area** the interpolation will be restricted to the xy-range of the actual picks. This may be useful if the picked surface does not cover the complete 3D-xy range. With deactivated option the interpolation range is identical to the complete xy-range of the actual 3D-file. A new pick will be generated by interpolation from the original picks for each gridpoint within the xy-range. 3 different **interpolation types** are available - linear, sqr and Gauss.

In addition 3 different **smoothing** modes are accessible. The resulting pick surface is smoother if the original picks are distributed reasonably uniformly in the x and y directions. If the actual picks exhibit an anisotropic distribution the parameter **n.pick** may be used in order to restrict the number of original picks and therefore to allow a more uniform distribution in x- and y-direction. The option **use pick code** controls if the pick code is taken into account. If activated only the picks with identical pick code form the base for one interpolation independently from the picks with other codes. Dependent on the number of different pick codes the total number of interpolated picks increases accordingly. The interpolated 3D-picks may be saved and loaded afterwards using the option view/picksurfaces for

PickInterpol3DGroupBox

n. pick: 1

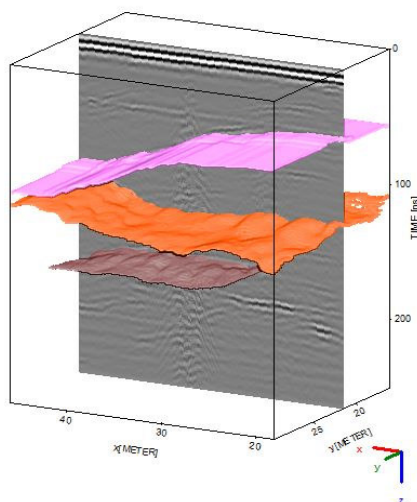
interpolation type:
☐ linear
☒ Gauss

smoothing:
☐ weak
☒ middle
☐ strong

☐ restrict to pick area
☐ use pick code

start cancel

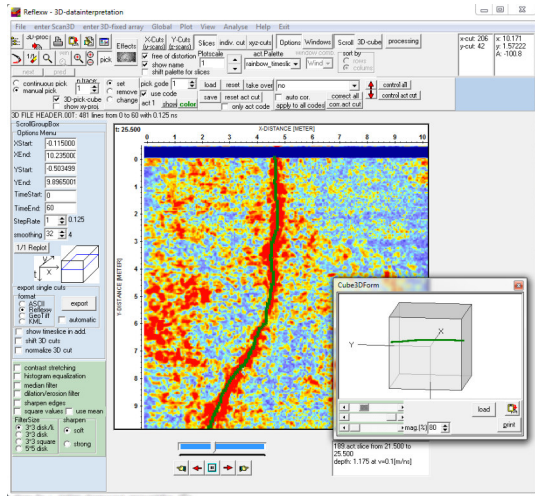
3D-cube. Here different 3D-pick datafiles for different layers (marked with different pick codes) or one 3D-pick datafile containing the picked layers within different pick code (see option use pick code).



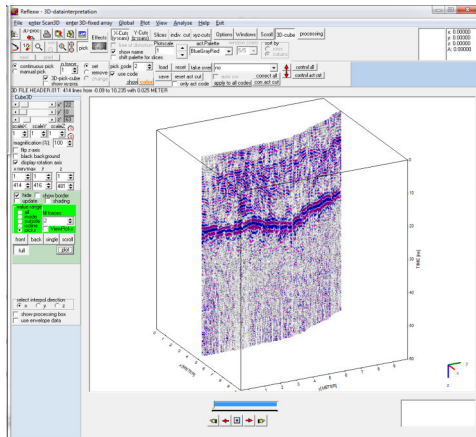
3.8.1 Picking a crooked target object with depth variations

Some new options allow to pick a crooked target object with a more or less linear structure and with depth variations (for example a pipe) semi automatically. For this purpose the following steps are necessary:

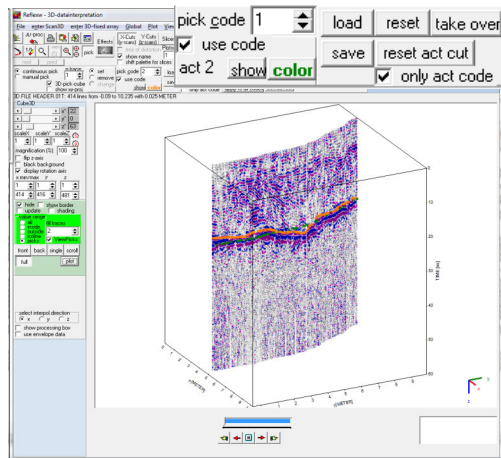
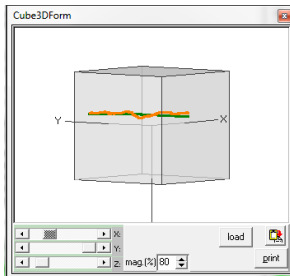
1. Choose **scroll** and activate **pick**. Choose an appropriate timeslice (a **smoothing** factor greater 1 might be useful if the target object differs in depth) and pick the object without consideration of the depth changes (see picture on the right) with activated option **use code** and using e.g. code 1. The result is the picked target showing the correct position in x- and y-direction but not in depth (see small picture of Cube3DForm, option 3D-pick-cube). Save the picks.



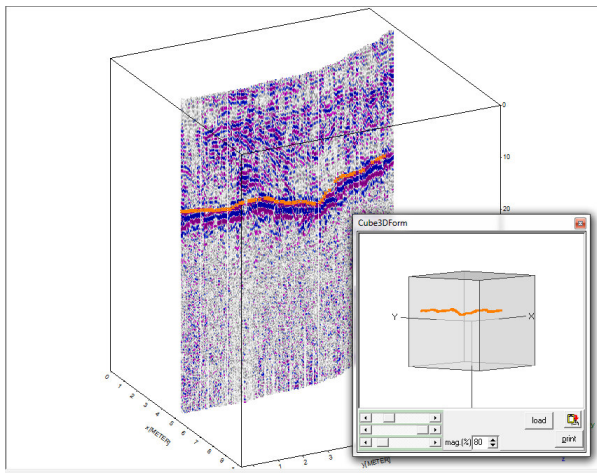
2. Activate **3D-cube** in order to only display the crooked line based on these picks. This is done by setting **value range** to **picks** and activate **full**. Now only these 3D-data traces will be plotted for which a pick has been set (picture on the right, the picks are not displayed if the option view picks is deactivated). The option **fill traces** allows you to avoid gaps because of missing pick traces. Enter here for example a value of 2 or 3 if gaps are present.



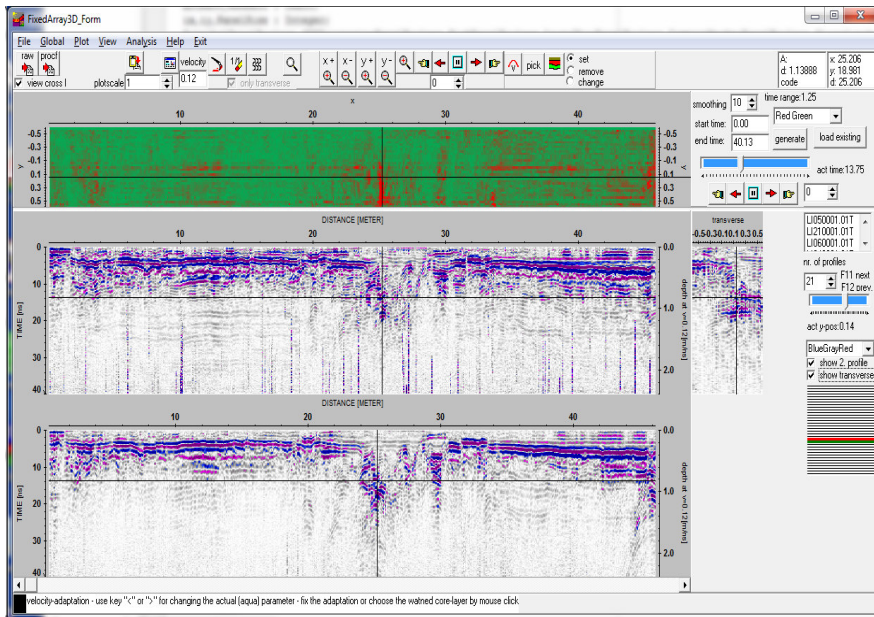
3. Now you are able to repick your target taking into account the depth changes. Change the pickcode to e.g. 2 and pick the pipe again using the right mouse button with e.g. activated option continuous pick. Now you have two pick sets with different code with the same xy coordinates but different depths (see Cube3DForm, picks in green and in orange).



4. For the last step you may delete the picks with the wrong depth by resetting pick code to 1, activate only act code and click on reset within the pick panel. Now only the picks with code 2 are present, save the picks.



3.9 3D-fixed array



The module allows the interactive interpretation of multichannel data with a fixed lateral offset of the individual channels based on relative coordinates. The target of this module is the interpretation of data consisting of many traces into one direction, e.g. along a road and a relatively small number of datapoints perpendicular to it (number of parallel lines). An easy and fast calculation of timeslices without interpolation but including smoothing in time direction is incorporated. A fast interactive switch between the different lines and timeslices is possible either using the mouse (pressed right mouse key or using the scroll wheel) or using the trackbars.

Precondition for the use is:

- identical timeincrement and number of samples of the individual 2D lines
- start time must be identical
- identical traceincrement and number of traces of the individual lines in profile direction
- the start coordinate in profile direction may be different for the individual 2D-lines
- energy normalization if necessary must have been performed before
- the filecoordinates must have been set correctly (start and end coordinates in profile direction and constant coordinate). This must have been done within a previous step, e.g. within the 2D-dataanalysis module.

The program uses the constant perpendicular coordinate (start and end should be identical) in order to sort the chosen datafiles after ascending coordinates

(perpendicular to the profile direction). The perpendicular coordinates do not need to be equidistant but the calculation of the timeslices does not take into account any different increments because no interpolation is done.

The max. number of parallel profiles is restricted to 128. The lines do not need to have been acquired within one step but these can even be acquired using a single channel system. The only need is that the coordinates have been set correctly before, e.g. within the 2D-dataanalysis.

In addition to these relative coordinates the traceheaders may contain the real, e.g. GPS-coordinates which may be distributed arbitrarily (see option view/profile line and view/gps map).

3.9.1 3D-fixed array MenuItems

The options within the uppermost panel allow a fast access to the data.



caption raw: open REFLEX raw data (see also File MenuItem)

The second button changes according to the last settings to

caption proc: open REFLEX processed data (see also File MenuItem)

caption proc: open REFLEX processed data with filter (see also File MenuItem)

The caption is automatically updated when using one of the following options within the File MenuItem: open rohdata, open procddata, open procddata filter, import, open Second Line/rawdata or procddata or procddata filter



copy to clipboard



enter PlotOptions

plotscale: enter multiplication factor for the color-amplitude assignment or for the wiggle size

velocity: enter a velocity for the transformation of the time scale to a depth scale for the depth axis and for the layer image.



replot current line with current zoom parameters



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



enable magnifying glass function



Zooming may be done by the manual zoom or by stepwise zoom in x- or y-direction (options x+, x-, y+ and y-). The zooming in time-(y-)direction is restricted to the upper part, this means the zoomed lines always starts at the overall start time. The timeslices will only

be zoomed in x- (profile) direction.

enable zoom - The option x+ increases the zoom in x-direction by a factor of 2. The option x- decreases the zoom in x-direction by a factor of 2. The x-zoom acts both on the profile display and on the display of the timeslice. The option y+ increases the zoom in y-(time)direction by a factor of 2. The option x- decreases the zoom in y-direction by a factor of 2. The y-zoom only acts on the profile display and not on the display of the timeslice. With the option ZOOM an arbitrary area of the data set can be selected and plotted in full screen size. The area to be enlarged, a rectangle, has to lie within a data set. Pressing the left mouse button

you determine a corner of this rectangle and by moving the mouse with pressed button the desired area is opened.



These options allow an automatic scrolling of the profile and the timeslice in x-direction if a zoom has been activated (scroll to the left, stop scrolling, scroll to the right). The spin button located beneath the autoscroll options defines a delay factor in order to slow down the scrolling. Value 0 no delay - value 500 max. delay.



enters the velocity adaptation MenuItem (see chap. 7.3).

pick button: Activating this option enters the Pick MenuItem (see chap. 7.4).



enter the LayerShow (see chap. 7.5).



the options set, remove and change control the picking and the interactive velocity adaptation

The profiles may also be displayed using the wiggle mode. The option **only transverse** restricts the wiggle display to the transverse profile.

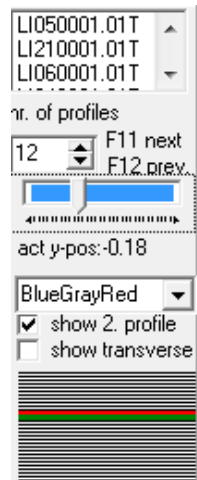
After having loaded the 2D-lines the chosen filenames will be displayed within the listbox using the order of the sorting after the profile constant coordinates. The 2D-lines are also displayed within an image. The actual 2D-lines is marked in red, a possible secondary line in green. The actual 2D-lines may be changed using the trackbar, the spin edit or the keys F11 and F12 as well as the right mouse button within the timeslice window.

The neighbouring parallel profile may be viewed in addition (option **show 2. Profile**) as well as the the actual transverse line (option **show tranverse**). Within this panel there is also the possibility of changing the color palette for the 2D-lines.

After having loaded the files and the timeslices (see chap. 7.2) there are several interactive possibilities to switch between the individual lines. The actual position within the profile and the timeslices and optionally within the 2. and tranverse profile are indicated by a large cross (option **view cross** activated).

A fast interactive change of the cuts is done by several mouse actions:

- the mouse wheel can be used in order to go through the lines or through the timeslices. For this purpose the right mouse button must be pressed once either within the timeslice or within the profile in order to set the focus to the timeslice or profile trackbar. The actual nr. of profile and the actual perpendicular position (act y-pos) will be displayed as well as the act. time for the timeslice.
- Moving the mouse with pressed right button allows a direct synchronization of the related profile or timeslice. This is only possible with act



The key F11 may be used to switch to the next profile and F12 to the previous one. This might be useful for picking within the 3D-picks mode (see below).

3.9.2 3D-fixed array timeslices

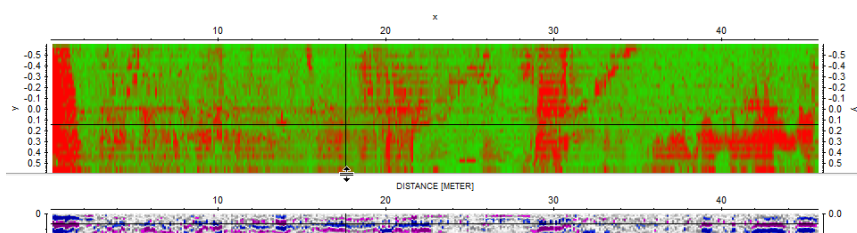
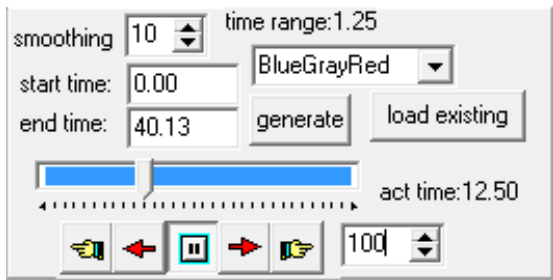
The **timeslices** may be created once by entering the start and end time and a smoothing parameter (option **generate**). The increment between successive slices is identical to the **timeincrement** multiplied by the **smoothing** factor. The number of slices depends on the start/end time and this increment. The timeslices do not need to be

generated for a further access if the same lines will be loaded for which the slices had been generated before. For this purpose the option **load existing** must be pressed after having loaded the parallel lines.

A **scrolling** through the timeslices may be manually done using the arrow bars: scroll one step to the lower or higher slices, autoscroll to smaller slices, stop scrolling, autoscroll to higher slices. A delay value may be entered in order to slow down the automatic scrolling.

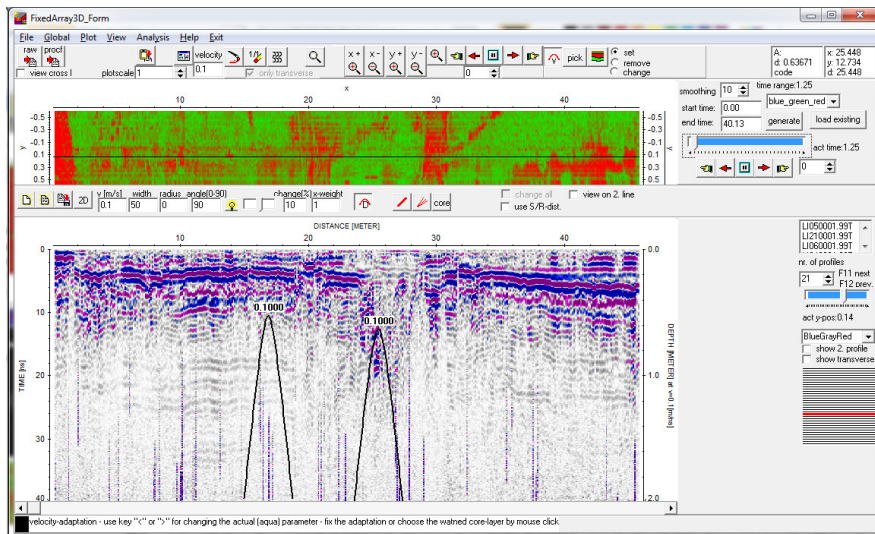
The color palette for the timeslices may be chosen independently from that used for the profile display.

The display size of the timeslice in relation to the display size of the 2D-profile may be changed using the splitter between the two images.



3.9.3 3D-fixed array velocity adaptation

The interactive velocity adaptation is analogous to that one of the 2D-datanalysis (see chap. 1.12.1) except the missing of reflection, intercept time and vsp adaptation. The left mouse button is used for the interactive velocity adaptation.



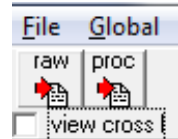
3.9.4 3D-fixed array picking

Picking is only possible within the working profile.



Normally the right button within the 3D-fixed array menu is reserved for moving between the different lines and slices. With the option **view cross** deactivated the right button is no longer reserved for these actions if the pick button is active. With these settings the following actions for the different pick methods are available using the right mouse button:

- manual pick: position. The right mouse button activates the interpolate option whereby a fast piecewise linear picking is possible.
- continuous pick The right mouse button allows you to toggle between the options set and change.
- phase follower The Escape-key as well as the right mouse button allows to abort the automatic assignment
- autopick: pressing the right mouse button within the working profile brings the autopick window to the front



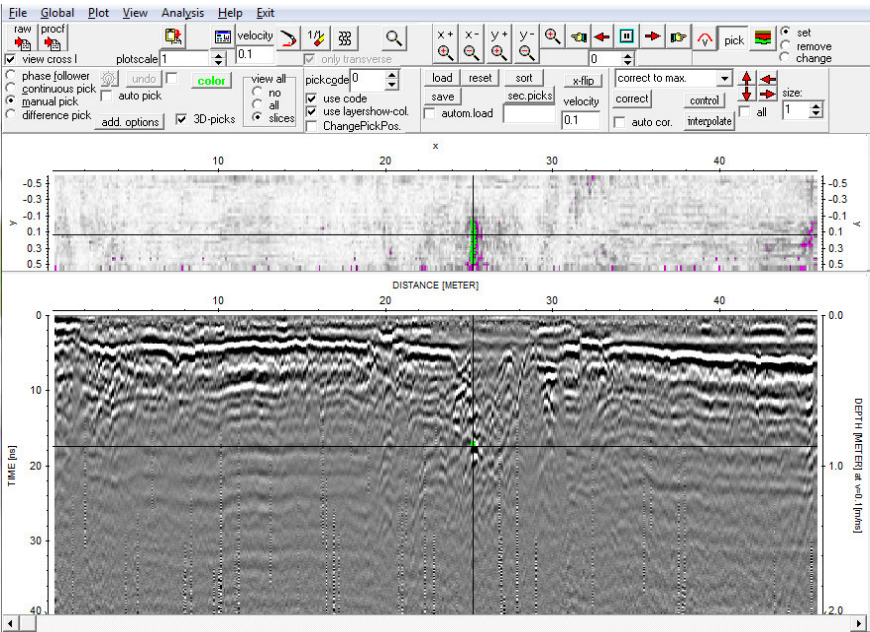
Two different picking methods are available which is controlled by the option **3D-picks**. First picking is done only for the actual working profile analogous to the 2D-dataanalysis. Second picking may be done for the whole set of profiles. For this purpose the option **3D-picks** must be activated.

3.9.4.1 2D-picking

With the option 3D-picks deactivated the picking is only done for the actual working profile. Almost all picking possibilities as described within the 2D-dataanalysis (see chap. 1.12.2) are available.

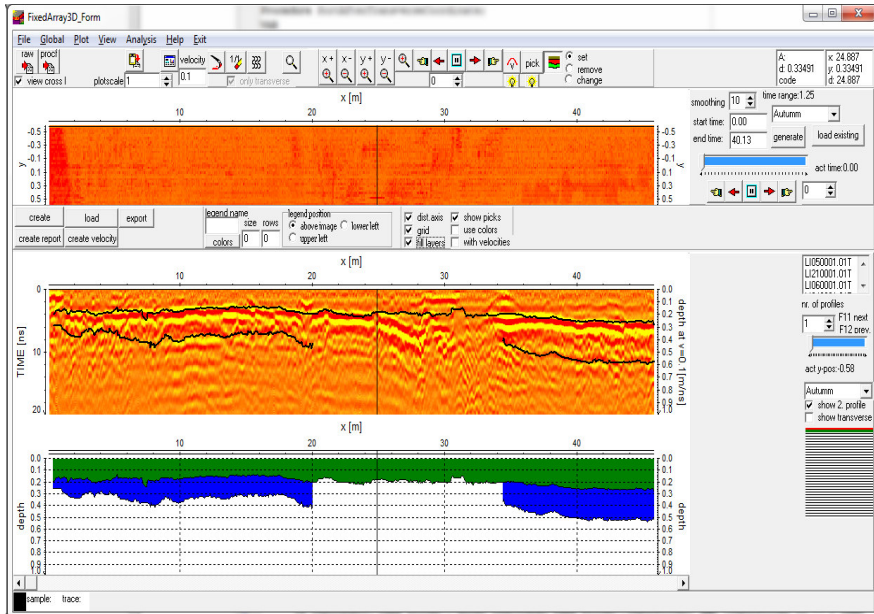
3.9.4.2 3D-picking

If this option is activated the picks are also displayed within the timeslice and only these picks belonging to the corresponding 2D-lines and to the actual timeslice will be displayed. Any change of the picks can only be done for the visible picks within the working profile window. The radio group **view all** allows you to display all picks either only within the timeslices (option slices active) or within the timeslices and the profiles (option all active). Analogous to the picking within the 2D-dataanalysis menu the code may be used for a distinction of picks belonging together. The keys F11 and F12 may be used for a fast switch between the different 2D-lines.

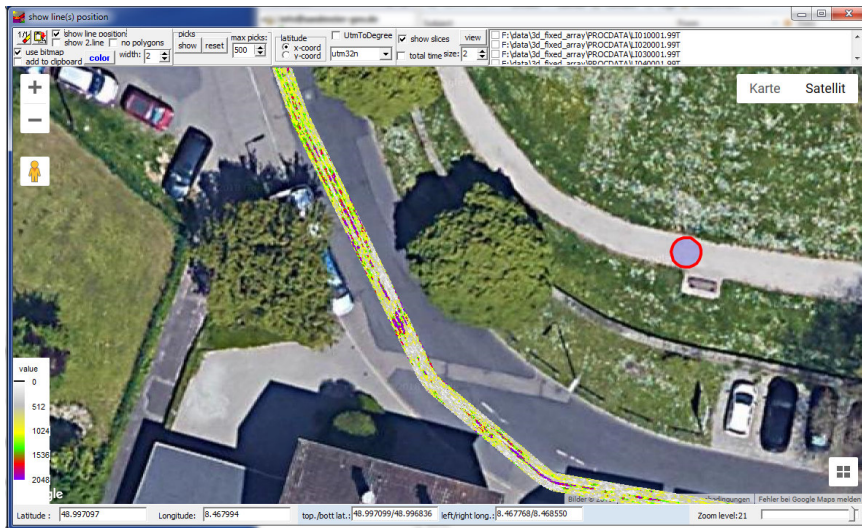


3.9.5 3D-fixed array layer show

The layer show options are analogous to those of the 2D-dataanalysis. The picks should have been generated for one single profile (pick option 3D-picks deactivated) - see chap. 2D-picking.



3.9.6 3D-fixed array gps map



Activating the option **view/gps map** allows to view the location of the profile lines and the actual timeslice within a GPS Map. The **color** and the **width** of the lines may be adapted. The display of the profile line may be disabled by setting width to 0. This might be useful for example if the pick display (see below) is hidden by the line. The display of the lines might be quite timeconsuming. Therefore the program automatically restricts the number of points per line to be displayed to 1000 points or if more than 10 lines (max.points 250) or more than 50 (max.points 100) lines are shown. Independently from these settings you also may restrict the number of points manually using the option **max.points**. With the option **show center** activated a center point of the line will be displayed in blue.

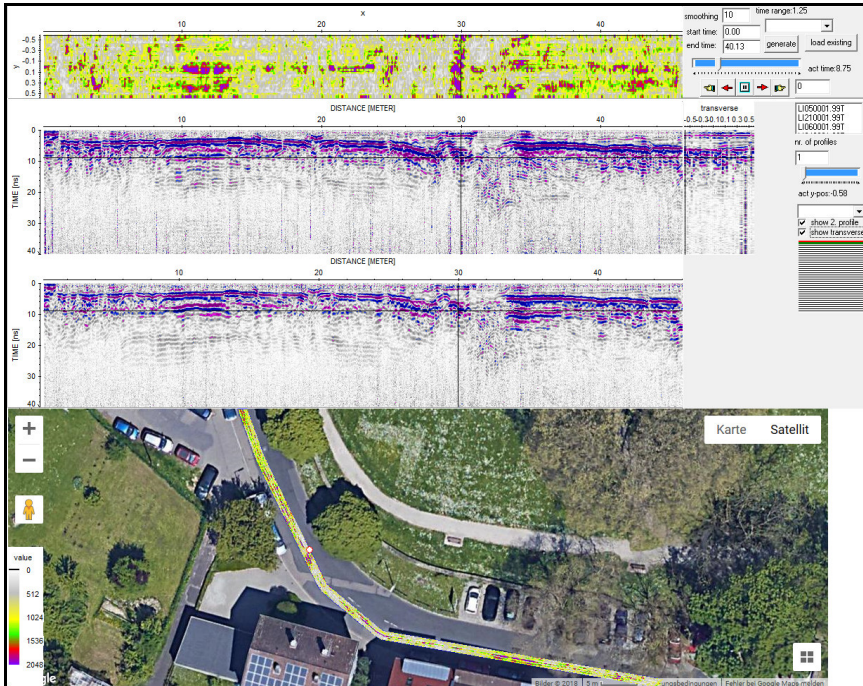
Activating the option **show line position** allows to continuously show the actual mouse position by a red circle when dragging over the line. The option **UTMToDegree** allows to convert UTM-coordinates to degree values based on a given utm zone. The option **latitude** defines where the latitude coordinates have been stored within the traceheaders (x- or y-position). **Zooming** and **copy to clipboard** possibilities are included. The option **1/1** resets and centers the display.

The display will also be automatically centered when using the zoom-option or when scrolling through the zoomed profile. The actual profile center will be displayed by a red circle.

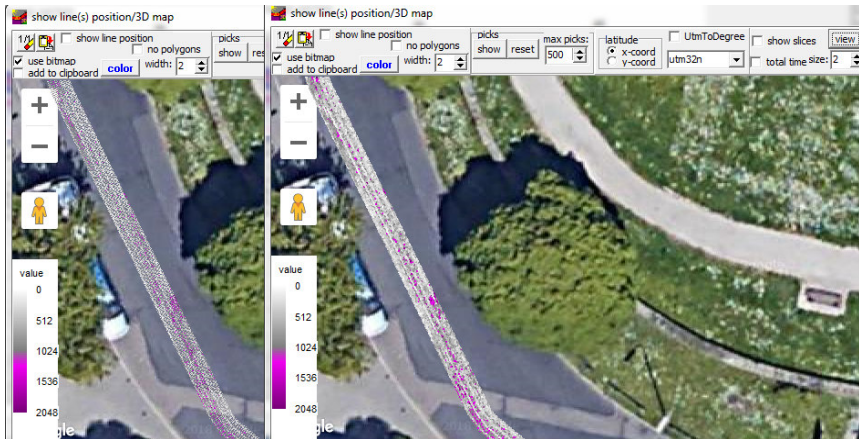
The actual **picks** may be displayed (option **show**) using a red-green-blue color scheme for the icons with red corresponding to the smallest traveltime values and blue to the largest traveltime values. With picks activated only the outer profile lines will be displayed. The parameter **max. picks** controls the max. picks to be displayed. If the actual number of picks exceeds this number only every n. pick will be displayed. **Reset** resets the pick display. The option **use bitmap** will be automatically activated and the color scheme for the pick traveltimes will be

displayed at the left lower border. With the pick option continuous pick activated the picks will be updated each time the continuous pick is interrupted (mouse button released).

add to clipboard: if activated the gpsmap will be added to the clipboard if started within the Fixed3DArray. For this purpose the option **use bitmap** must be active. This option generates an internal bitmap copy of the gps map which is necessary for any superposition of other bitmaps like the actual timeslice.



The actual **slice** may also be displayed within the gps map (option **view** within the slice panel). The option **use bitmap** will be activated automatically. The slice will be displayed using the traceheader coordiantes. Each datapoint will be represented by one pixel. The option **size** allows to increase the size of the display. This is necessary if the zoom level is too high in order to ensure a complete filling.

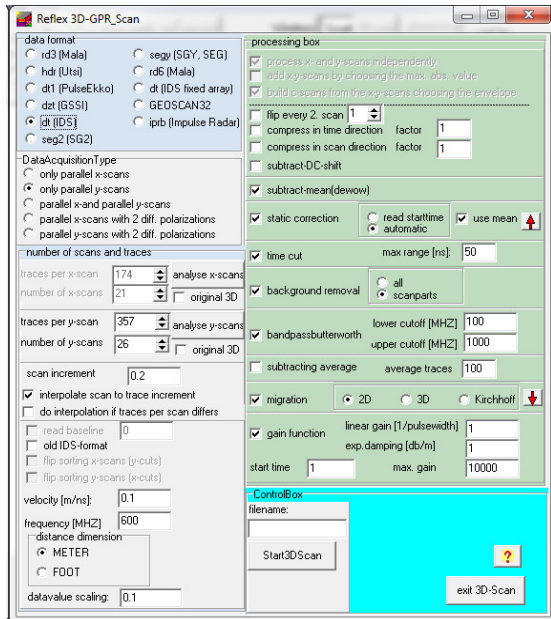


3.10 Scan 3D

The Scan 3D tool allows to import and to analyse automatically rectangular 3-dimensional GPR-data which have been acquired along 2D-parallel lines in one or two perpendicular directions.

2D-lines on a regular rectangular grid. This means that the traceincrement in one direction (x or y), the startposition of the 2D-lines (exception IDS fixed array which allows different start positions) and the scan increment between the 2D-lines must be equal. The data may have been stored on individual 2D-files or within a 3D-file. If stored within a 3D-file the end-positions of the internal 2D-lines must be identical in addition as well as the number of traces into profile direction.

The **traceincrement** must be given within the original data. In addition if the data have been acquired in two perpendicular directions (**crossing lines**) the start- and end-positions of the 2D-lines must be identical and the trace increment must be equal for all 2D-lines but the scan increment between the parallel lines does not need to equal the traceincrement if the option interpolate scan to trace increment is active. In this case an **automatic interpolation** will be done. If these preconditions are satisfied the 3D-scan program allows a very fast interpretation of your 3D-data.



The program supports the

dataformats of the following GPR-manufactures:

Mala (rd3 and rd6 files), Utsi (hdr files), PulseEKKO (dt1 files), GSSI (dzt files), IDS (dt files), Geotech (GEOSCAN 32, gpr2 files) and ImpulseRadar (iprb files). In addition the SEG2- and the SEG1-dataformat are supported. The dataformat must be specified within the **data format box**.

For both the SEG2 and the SEG1-format the data are assumed to be in ns. In the case of SEG2-data the timeincrement is assumed to be in ps.

The **rd6 (Mala)** format is used with the MALA CX system for a 3D-grid data acquisition. Three different datafiles must be present: rhd, rad and rd6. The rd6 file must be chosen when starting the conversion - otherwise a warning message appears. The rad-file corresponds to the rad-file of the Mala rd3-format and contains informations about the data acquisition like sampling rate, The rhd-file contains informations about the 3D-grid like the x-size and y-size, the resolution in profile direction and the profile spacing. One or two rd6 files are present. They have the same filename like the rad and rhd file but in addition '_1' for the grid lines in y-direction and '_2' for the grid lines in x-direction. The 2D-lines building up the 3D-grid are sequentially stored corresponding to the Reflex3D-file. If only one dataset is chosen (option only parallel x-scans or only parallel y-scans active) the program controls the setting and changes it automatically if necessary corresponding to the choose of the rd6 file (only parallel y-scans for file <name>_1.rd6 and only parallel x-scans for file <name>_2.rd6) .

In most cases the profile spacing (perpendicular to the profile direction) is larger than the spacing in profile direction. Therefore the program automatically performs an interpolation if both the parallel x- and y-scans are chosen in order to ensure the same grid resolution in both directions (option interpolate scan to trace increment is automatically activated). If only parallel x-cuts or parallel y-scans has been chosen such an interpolation will not be done.

Example: x-size: 0.8 m, y-size: 0.8 m, spacing in profile direction: 0.004 m, spacing perpendicular: 0.1 m.

Only parallel x-scans: the resulting 3D-file consists of 201*9 points (9 x-cuts with each cut containing 201 traces)

Parallel x- and y-scans: the resulting 3D-file consists of 201*201 points

The number of traces and scans and the scan increment do not need to be entered for the rd6-format as these informations will be read from the rhd-file.

The options **flip sorting x-scans (y-cuts)** and **flip sorting y-scans (x-cuts)** have been introduced because it may happen dependent on the type for the CX system that the data have not been stored sequentially scan (channel) after scan (channel) but first trace of all x channels, then second trace of all x channels and so on. . If this holds true you must activate the corresponding option for the x-scans and/or for the y-scans.

The **dt (IDS fixed array)** format is used with the IDS RIS or RIS Hi-BrigHT or STREAM X system for a 3D-grid data acquisition. The original data of these systems (parallel 2D-lines) contain the relative positions of the antennas to the origin of the coordinate system. The complete geometry is taken from these coordinates. Therefore it is not necessary to input the number of scans and of traces. In addition the startcoordinate of each 2D-line may vary. The IDS naming is the following: filename+channelnumber+running number of the scanarray. Example: LI0800001.dt means channel number 8 of the first scan array. Some arrays include two different antennas with two different polarizations, for example RIS Hi-BrigHT with 8 VV antennas for the first 8 channels and 8 HH antennas for channel 9-16.

The file sorting depends on the given antenna geometry perpendicular to the scan direction. Therefore it is also possible to acquire different interlaced scan arrays in order to decrease the profile increment perpendicular to the scan direction but you must care that the resulting profileincrement between adjacent scans must be the same.

Example: the antennas of the RIS Hi-BrigHT system have a distance of 0.1 m beginning at 0.1 m from the border of the the array block. It is possible to acquire a second scan array beginning at 0.15 m from the border of the the array block in order to increase the resolution perpendicular to the scan direction (resulting profileincrement: 0.05 m).

The antennas also have a shift in profiledirection which is automatically taken into account.

At present for the IDS fixed array the DataAcquisitionType parallel x and parallel y-scans is not supported. For crossing lines the data in x-direction (type only parallel x-scans) and y-direction (type only parallel y-scans) must be processed separately and stored on a file (a filename must be entered). The two 3D-files can be combined later within the 3DGPRScansView module using the option load second file.

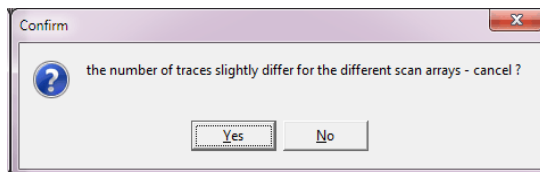
If two different polarizations are available this shift is normally different for the two polarizations. If you want to combine the two polarization scans you have two possibilities:

1. Use DataAcquisitionType parallel x-scans with 2 diff. polarizations or parallel y-scans with 2 diff. polarizations: then you must choose all data and the data must have the file naming as mentioned above. Again different scan arrays can be loaded within one step. The positions perpendicular to the scan direction must be the same for the two polarization directions. This holds true e.g. for the RIS Hi-BrigHT array. With the option process x-and y-scans independently activated all data for one polarization are

processed together. The different shifts in profile direction are automatically taken into account.

2. Use DataAcquisitionType parallel x-scans and choose the data for only one polarization. Enter any filename, e.g. polar1 for the first polarization and build the 3D-datafile. Then enter another filename, e.g. polar2 for the second polarization and build a second 3D-datafile. Within the 3D-dataintepretation module you have the possibility to load a second 3D-datafile. The message for the automatic shift should be confirmed. You can either show the second datafile within a second window or you may overlay it which gives the same result as using the first possibility.

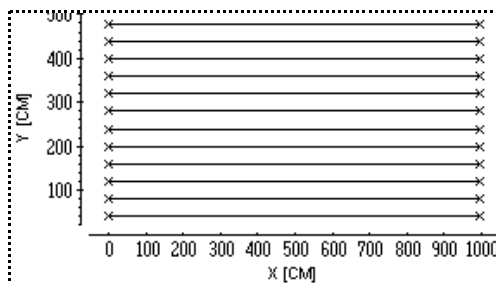
The following message may appear if different scan arrays have been chosen. It may be ignored if the number of traces for the different arrays does not differ too much.



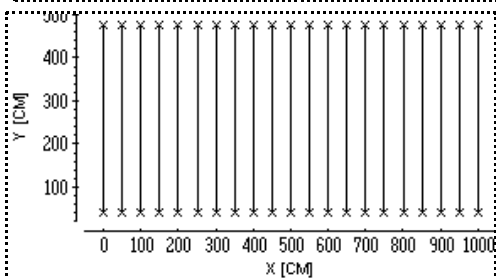
The **GEOSCAN 32** format (.gpr and .gpr2 extension) uses data from the OKO-2 GPR of Geotech. The trace increment is read from the original header. No start coordinates will be taken into account.

It is possible to analyse parallel lines in one direction or two perpendicular directions. This has to be specified within the **DataAcquistionType** box.

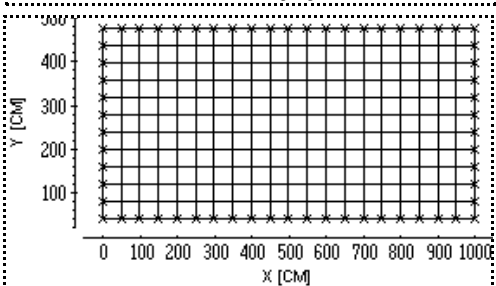
Choose *only parallel x-scans* if you only have acquired parallel lines in x-direction.



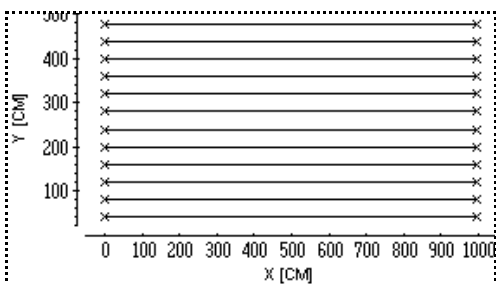
Choose *only parallel y-scans* if you only have acquired parallel lines in y-direction.



Choose *parallel x- and parallel y-scans* if you have acquired both parallel lines in x-direction and y-direction.



Choose *parallel x-scans with 2 diff. polarizations* if you have acquired parallel lines in x-direction with 2 different antenna polarizations - the first one perpendicular to the scan direction (equal y) and the other one parallel to the scan direction (equal x).



The original data may be stored on individual 2D-files or on one 3D-file with the 2D-lines sequentially stored.

To be considered: The 3D-data file must start with the x-scans at the 0-position until the last position of the x-scans and then the y-scans follow.

If the data have been acquired on different files the program automatically sorts the files after the filenaming in alphabetical order. Therefore the naming of the individual filenames must have been done in such way that the above sorting condition holds true.

Example 1: 2 3D-files: filex.rd3 (parallel x-scans) and filey.rd3 (parallel y-scans). The sorting condition holds true because filex.rd3 will be the first file after the filename sorting.

Example 2: 20 2D-files in x-direction and 15 2D-files in y-direction: file01x.rd3 - file20x.rd3 and file01y.rd3 - file15y.rd3. Again the sorting condition holds true.

Depending on the chosen dataacquisition type you have to specify the number of scans and traces in x- and/or y-direction within the **number of scans and traces box**. The option analyse x-scans or analyse y-scans allows to analyse the number of traces within the original data. After having pressed the option you have to choose all the original datafiles containing the x- or y-scans. The number of choosen datafiles, the total number of traces and the average number of traces will be displayed. With the option original 3D activated the original file is already a 3D-file containing all data. The program tries to find the individual 2D-scans within the original 3D-file and shows the result as markers together with 3D-profile for a visual verification. This finding only works if the individual 2D-scans exhibit a remarkable change so that the finding algorithm is able to locate the positions. Within the number of scans and traces box you also may specify the mean velocity of the underground (necessary for example for the migration and for the depth axis), the nominal frequency (necessary for example for the static correction) and the scan increment perpendicular to the scan direction. The traceincrement will be read from the original data.

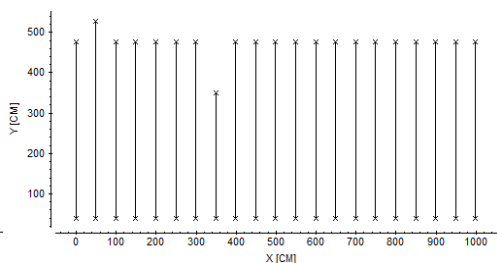
If both x- and y-scans have been acquired the scanincrement must be equal to the traceincrement or the option interpolate scan to trace increment must be activated. If activated the program automatically performs an interpolation of the data perpendicular to the profile direction based on the entered scan increment and the given traceincrement. The number of 2D-lines building up the 3D-file changes accordingly. The scanincrement should be a multiple of the traceincrement. Precondition for the case of xy-data acquisition is that the entered number of scans - 1 is a multiple of the entered number of traces -1 in perpendicular direction. In addition the grid size defined by the number of x- and y-scans and the scan increment should fit size given by the number of traces in x- and y-direction and the traceincrement. The program automatically controls this size and gives a warning message if no fitting is given and asks for an automatic change of the number of traces in x- and y-direction.

The program automatically controls if the total number of traces of the original data equals the entered total number (traces per x-scan*number of x-scans + traces per y-scan*number of y-scans). If the numbers differ a warning message appears and you may cancel the interpretation or not. There might be different causes for the difference:

1. The entered number of scans and traces is wrong - please cancel the process and enter the correct numbers.
2. The data have been stored on individual 2D-files and the number of traces per scan is not exactly the same for all 2D-lines although a wheel has been used. In this case a reinterpolation of the individual 2D- lines might be okay in order to compensate for the errors. The program automatically performs such an interpolation (option do interpolation if traces per scan differs activated) if the

process has not been cancelled by the interpreter. This is possible if the original data have been stored on individual 2D-files because in this case the endposition of each scan is fixed.

3. The data have been stored on individual 2D-files and the number of traces per scan is not exactly the same for all 2D-lines because the endpositions are different (see picture on the right). In this case deactivate the option do interpolation if traces per scan differs. The program determines the max. number of traces which occurs within all loaded 2D-scans and adds zero traces at the end of those scans which exhibit a lower number of traces.



4. The data have been stored on a 3D-file and the number of traces per scan is not exactly the same for all 2D-lines within the 3D-file although a wheel has been used. In this case no reinterpolation of the individual scans can be done because the endpositions of the 2D-scans are not known. Therefore cancelling of the process is recommended.

The **distance dimension** may be either METER or FOOT.

Optionally some filter steps are automatically performed. These must be defined within the

ProcessingBox.

Dependent on the original data different processing steps must be chosen. The data from Mala, Utsi, PulseEkko and IDS are normally raw data, this means no analog gain or filter has been applied during the data acquisition.

Therefore the dewow filter and the gain should always be used. Often a static correction is useful for all dataformats and the migration should be used if diffractions are present. The background removal may significantly increase the data quality but in some cases the opposite effect may happen. Therefore there is no general rule if and how to use this filter. The sequence of the filters cannot be changed except the order of 'static correction' and 'subtract mean (dewow)'. In this case the red arrow button can be used in order to change the sequence.

processing box	
<input checked="" type="checkbox"/>	process x- and y-scans independently
<input type="checkbox"/>	add x-y-scans by choosing the max. abs. value
<input checked="" type="checkbox"/>	build c-scans from the x-y-scans choosing the envelope
<input type="checkbox"/>	flip every 2. scan <input type="text" value="1"/>
<input type="checkbox"/>	compress in time direction factor <input type="text" value="1"/>
<input type="checkbox"/>	compress in scan direction factor <input type="text" value="1"/>
<input type="checkbox"/>	subtract-DC-shift
<input checked="" type="checkbox"/>	subtract-mean(dewow)
<input checked="" type="checkbox"/>	static correction <input type="radio"/> read starttime <input checked="" type="radio"/> use mean
<input checked="" type="checkbox"/>	time cut max range [ns]: <input type="text" value="50"/>
<input checked="" type="checkbox"/>	background removal <input type="radio"/> all <input checked="" type="radio"/> scanparts
<input checked="" type="checkbox"/>	bandpassbutterworth lower cutoff [MHZ] <input type="text" value="100"/> upper cutoff [MHZ] <input type="text" value="1000"/>
<input type="checkbox"/>	subtracting average average traces <input type="text" value="100"/>
<input checked="" type="checkbox"/>	migration <input checked="" type="radio"/> 2D <input type="radio"/> 3D <input type="radio"/> Kirchhoff
<input type="checkbox"/>	no migration for xy-cuts
<input checked="" type="checkbox"/>	gain function linear gain [1/pulsewidth] <input type="text" value="1"/> exp.damping [db/m] <input type="text" value="1"/>
start time <input type="text" value="1"/>	max. gain <input type="text" value="10000"/>

Changing this order may lead to better results of the static correction filter.

flip every 2. scan: With this option activated every 2. scan will be flipped in scan direction. This allows you to process and interpret data which have been acquired in a meandering manner (forth and back).

compress in time direction: With this option the data in time direction will be compressed by the entered factor.

compress in scan direction: With this option the data in scan direction will be compressed by the entered factor.

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

Seismic shot data often reveal such a DC-shift which will be best eliminated using this filter.

static correction: With this option activated a static, this means a time-independent correction for each trace in time direction will be done. The time shift will either be read from the original data (option read starttime activated) or will be automatically determined from the data (option automatic activated). The data above the time shift level will be lost.

With the suboption automatic activated the timeshifts are automatically determined. For that purpose the first significant first arrival (the amplitudes must exceed 10 % of the max. value within the trace) will be searched which gives the timeshifts for each trace. With the option use mean activated only the mean time shift is used for all traces of a 2D-subdataset.

subtract mean (dewow): With this option activated a running mean value is calculated for each value of each trace. This running mean is subtracted from the central point. The time window for the calculation of the running mean value is calculated from the entered frequency given in MHz. This filter may be used for eliminating a possible low frequency part (dewow). The filter may not be enabled if a high pass filter has already been applied during the data acquisition.

time cut: With this option activated the length of each trace may be limited to a predefinable maximum time which has to be specified within the parameter max.time in the given time dimension.

background removal: This filter performs a subtracting of an averaged trace, a so called background removal. With this option you can eliminate temporally consistent noise from the whole profile and therefore possibly make signals visible, previously covered by this noise. This filter method also suppresses horizontally coherent energy. Its effect is also to emphasize signals which vary laterally (e.g. diffractions).

The average trace will be built either from the complete 3D-data (suboption all activated) or for each 2D-line within the 3D-datavolume separately (suboption scanparts activated) over the complete timerange.

attention: it might happen that the filter causes non real signals. This holds true when the averaged time series contains energy which is not present within any part(s) of the profile.

bandpassbutterworth: Here you can apply a bandpass filtering in the frequency domain using a butterworth taper of order 2. The filter band is specified by the setting of two frequency values. The first point determines the lower cutoff frequency, the second one the higher cutoff frequency. The frequency spectrum

below the low cut and above the high cut frequency is tapered using a butterworth taper of the order 2. By the corresponding choice of the points of the bandpass either a lowpass or a highpass can be approximately realized. Noise can be suppressed with the bandpass filter when it differs from the signal in its frequency content.

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

migration: A 2D or 3D fast fk-migration on the basis of a constant velocity is performed. The profile must represent a so called zero-offset profile, i.e. shot and receiver have to be at the same location. The goal of the migration is to trace back the reflection and diffraction energy to their "source". A zero offset section often does not represent the "true" position of the reflectors mainly for steep layers. After the migration often a better approximation to the reality is given. If strong diffractions are present the migration tries to contract these diffractions to a minimum. This is useful for an interpretation using timeslices for example. The method works in the frequency-wavenumber (fk) range (see also fk spectrum). Within the fk-range a variable transform is done based on the entered constant velocity (frequency is transformed onto the vertical wavenumber). First the x-t data are transformed into the f-k-range. After having done the transformation the migration process will be done and after that the back transformation into the x-t-range is performed. The option **no migration for xy-cuts** allows to restrict the migration to the timeslices. This may be helpful if both the xy-cuts and the timeslices are displayed using for example the option xyz-cuts.

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

The data are multiplied by this function in order to compensate for possible damping or geometric spreading losses.

If you have acquired both x- and y-scans you also may choose if the x- and y-scans may be processed independently and how the adding of the x- and y-scans shall be done. Activating the option **add x-y scans by choosing the max. abs. value** means that the x- and y-scans are not simply added but the max. absolute

value will be determined and this value is taken as the sum value. This has the advantage that no information will be lost when the two different scans are added. Activating the option ***build c-scans from the xy-scans choosing the envelope*** means that the c-scans(timeslices) are built independently choosing the envelope of the original data.

The program automatically creates some datafiles under the working directory for storing the intermediate results. They are Scan3DFile.dat, Scan3DFileSec.Dat, Scan3DFileX.00t and Scand3DFileY.00t. These datafiles are always overwritten. By default the **working directory** is the program directory. You may change the working directory (option manual directory activated). This setting is stored within Reflex3DScan Ini-File.

After having started the 3D-scan program (option **start3DScan**) the original data are queried (multiple file choice using the key str of shift). The default path for the original data is the actual projectdirectory but you may load the data from any other path. Then the import, data sorting and processing will be done automatically and the **3D-GPR_ScanView** module opens. By default the Reflexw 3D-data are stored under the filename Scan3DFile.dat under the actual projectdirectory.

You may specify the **filename** of the resulting 3D-file. For the case of a data acquisition of both x- and y-scans and activated option ***build c-scans from the xy-scans choosing the envelope*** a second 3D-file will be stored with the same filename and an additional nameextension SEC.

4. Modelling

The menu modelling allows the interactive production of a 2-dimensional layer model and based on this layer model the simulation of the electromagnetic and seismic wave propagation by means of finite difference (FD) methods or ray tracing. In addition, a tomographic approach is included, which allows the automatic adaptation of transmitted travel time data.

The possibilities of the menu modelling as a summary:

- interactive production of a 2D-layer model with given physical parameters (relative dielectric permittivity ϵ [], relative magnetic permeability μ [] and electric conductivity σ [S/m] or pressure wave velocity V_p [m/s], shear wave velocity V_s [m/s] and density ρ [g/cm³], respectively)
- simulation of electromagnetic, seismic wave propagation, respectively by means of **FD-method** for different sources (plane wave, point source as well as "Exploding-Reflector"-source), storage of the complete wave field on disk
- generating of arbitrary profiles, snapshot sequences, respectively, from the calculated wave field
- **tomographic inversion** of transmission travel time data
- calculation of **synthetic traveltimes** using two different methods (see raytracing 2D, chap. 4.7.1) and to invert picked forward and reciprocal traveltimes using a **wavefront inversion technique** (see wavefront-inversion 2D, chap. 4.7.2).

The menu modelling allows the interactive production of an arbitrary **2-dimensional layer model** (e. g. wedging-out interfaces). The distance dimension is either METER or FOOT (must be specified within the global settings menu, chap. 1.2.1). The FD-modelling of electromagnetic waves only allows the distance dimension METER. The max. number of layers is restricted to 500. It is possible to generate isolated elements as well as to fall back on predefined symbols (circles, rectangles, triangles or lenses). A topography can be given, too. A layer interface can be made of a maximum of 250 different x,z-points. Older Reflexw versions (up to 7.5) only allow 100 x,z-points. Different physical parameters can be assigned to any layer point, so that the physical parameters of a layer can vary laterally. In between the entered values linear interpolation in the x-direction occurs. A set of parameters is valid for the whole depth area (z-axis) below the current interface. The interfaces do not have to be entered exactly at the edge, intersection, respectively, but only in the vicinity because the option **extrapolate** allows the automatic interpolation between two interfaces as well as the extrapolation of the interface at the border. 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). This is necessary for the following FD-rastering.

The option **hor.extrap.** allows the extrapolation of the interface only to the left and right borders of the model using a horizontal line. Use this option instead of extrapolate if you want the interface to be only extrapolated to the vertical borders. The option **lin.extrap.** allows the extrapolation of the interface only to the left and right borders of the model by using the slope of the first/last model points. Use

this option instead of extrapolate if you want the interface to be only extrapolated to the vertical borders.

On the basis of an arbitrary 2-dimensional model a **finite difference-simulation** (FD-computation) of the electromagnetic, seismic wave propagation, respectively, can be executed. The result of this calculation is the wave field depending on x , z and t (time). This method allows the simulation of the kinematic (travel time) as well as dynamic (amplitude, frequency etc.). An explicit method is used, i.e. the wave field is calculated for a certain point of time with the values of the preceding point of time etc.. The layer model has to be rastered for the calculation with a given increment in x - and z - direction. Just so an increment for the time has to be given. The size of the space-increment is in accordance with the minimal wave length, i.e. it depends on the choice of the physical parameter as well as the given main frequency. If the increment is given too coarse, numerical dispersion appears, i.e. the propagation velocity depends on frequency even if the physical parameters are given frequency independent. The program determines automatically the critical value of the space-increment and shows this one in a window. This value should not be passed over. The max. time-increment depends on the max. velocity V as well as on the given space-increment Δx (approximately: $\Delta t \leq \Delta x / (\sqrt{2}V)$ for the electromagnetic propagation and $\Delta t \leq \Delta x / (V_p + V_s)$ for the elastic propagation). In the case of the electromagnetic wave propagation and a high conductivity the critical time value can be dominated additionally by the condition: $\Delta t \leq \epsilon / \sigma$. If one of these critical values is passed over the FD-scheme turns to be instable, i.e. the calculated values increase continuously with time and under certain circumstances an overflow is possible, i.e. the values cannot be displayed binarily. The critical time-increment is displayed in the same window as the critical space-increment. Moreover, the critical time-increment for the currently set space-increment is shown. The computation time, necessary for the FD-computation, depends linearly on the number of time points as well as on the number of space points in x - and z -direction, i.e. decreasing the space-increment by a factor of 2 increases the computation time by a factor of 4 (currently by a factor of 8, because the time-increment has to be decreased, likewise). The electromagnetic simulation is based on the solution of the Maxwell-equations. By doing so, it is assumed that the physical parameters ϵ , μ and σ are frequency independent and that the physical parameters are constant in y -direction. In each case the x -, y -, z -components of the electrical (E) and magnetical (H) field are calculated. If the source excites only in E_y -direction, in H_x - or H_z -direction, respectively, the needed computation time decreases by a factor 2 because the other fields are decoupled and are identical to 0 in this case (TE-Mode). The seismic simulation is based either on the 2-dimensional elastic-, acoustic- or SH (shear wave, polarized in y -direction) -wave equation according to the chosen method. In all cases no attenuation is taken into consideration and the elastic parameters are assumed to be frequency independent. The computation for the acoustic case is much faster than for the elastic case. In the case of the elastic computation the vertical (z) and the horizontal (x) displacement is computed. The potential is calculated in the case of the acoustic and SH- modelling.

The model-boundaries can be reflective or absorbing. In the case of electromagnetic wave propagation the model-boundaries can be enlarged by an absorbing edge area, in the case of seismic propagation an absorbing boundary condition is set on.

Three different sources are supplied. The plane wave can be excited in an arbitrary depth level z . It allows, e. g. the simulation of a zero-offset-section. But

here it has to be noticed that the reflection characteristics are only equivalent to those of a zero-offset-section in the case of horizontal plane reflectors. Additionally, it has to be noticed that a loss of divergence, as it happens in a field experiment, where single events are put together to a zero-offset-section, can't be considered for a plane wave excitation and thus the amplitudes reflect the reality limited. The point-source can be placed at any arbitrary point in the 2D-model. By it, a single experiment can be simulated kinematically and dynamically correct (e.g. a CMP-experiment or a shot-section). Several single calculations can be put together to a zero- or constant-offset section (very computation time intensive), too. Anew, the exploding-reflector-model allows the simulation of a zero-offset-section in a single FD-simulation. Using the exploding-reflector-model means that starting at time $t=0$ all points stemming from a reflector are the starting point of a Huygen's elementary wave with an amplitude proportional to the reflexion coefficient for the normal incidence case. During the rastering the source strengths are calculated and stored on a special file. For the electromagnetic case the phase is calculated in addition. For the exploding-reflector-model, contrary to the plane wave, reflectors which are not plane are described kinematically correct. In regard to the amplitude characteristic the same is valid as for the plane wave, which impinges on a plane reflector.

The FD-program is an autonomous program (program name FDEMSEIS.EXE) and is automatically started from REFLEXW.

After terminating the FD-computation one or several single lines or the complete wave field, depending on the space coordinates x and z as well as on time t , are stored automatically in the directory ROHDATA of the current project directory. Thereby it can be given which component has to be saved (electromagnetic wave propagation: selecting out of E_x , E_y , E_z , H_x , H_y , H_z ; elastic wave propagation: vertical or horizontal displacement). Additionally, the sampling in x , z as well as in t can be selected coarser in order to minimize disc space. The data are stored in the REFLEXW-internal format and can be viewed and processed like every other data profile.

Use the speed buttons in the first lower panel on the left for a speed access to some further different options:



enter the project directory



copy model to the clipboard



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



enable manual zoom - With the option ZOOM an arbitrary area of the data set can be selected and plotted in full screen size



reset all model parameters - All model parameters apart from the dimensions are reset



load model from file



Save model on file



Print model

- FD-Button:** FDGroupBox is activated - In this Box all the parameters for the FD-computation are set
- Tomo-Button:** TomographyGroupBox is activated - In this Box all the parameters for the tomography are set
- ray-Button:** RayGroupBox is activated - In this Box all the parameters for the raytracing/wavefront inversion are set

Within the **Main Menu Bar** you may choose the following items:

File Menuitem: From this menu you may choose your project directory, load, save, reset and print an existing model and copy the current model to the clipboard.

Edit Menuitem: Perform editing functions (e.g. undo-function).

Global Menuitem: Enter some global settings like fonts, printer,... (For further information refer to Global Menuitem in chapter 1.)

View Menuitem: This item controls the display options.

Plot Menuitem: Within this icon you may activate all the plot settings. (For further information refer to Plot Menuitem and Plot Options in chapter 1.)

The option **Reset traveltimes/rays** resets the display of the loaded traveltimes (real and simulated) and displays only the model.

The option **Resize based on free of distortion** changes the window size in such a way that the x- and z-axis of the model are free of distortion.

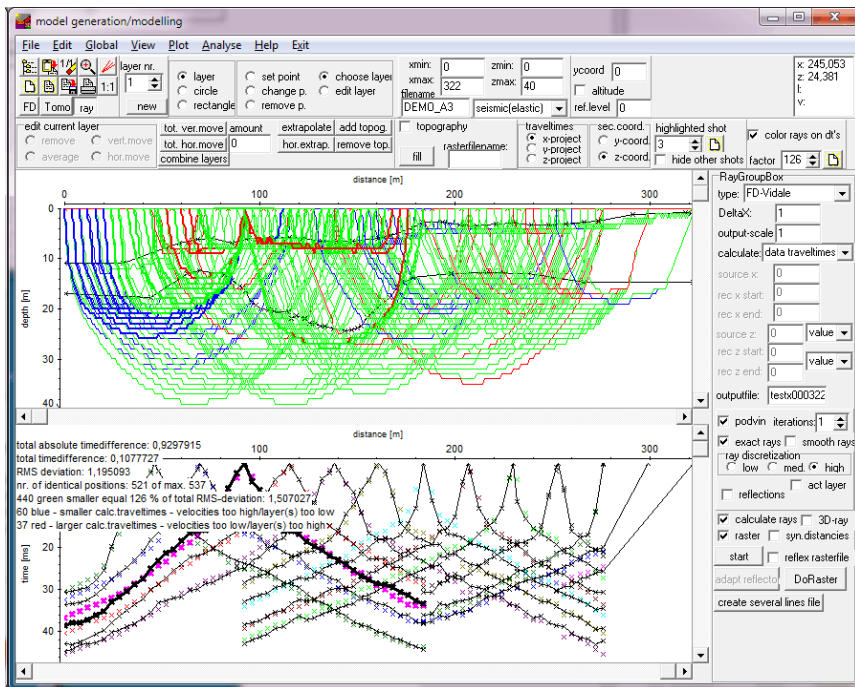
Analyse Menuitem: Enter the wanted analysis option FD, Tomo or ray (see above).

The options **calculate travelttime differences using actual coord.projection** and **calculate travelttime differences using all coordinates** allow to calculate travelttime differences between the data and the calculated traveltimes. 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 determining the quasi-identical positions. By default the current raster increment of the raytracing is used. The two options differ in the method to determine the common positions. If calculate travelttime differences using actual coord.projection has been chosen only the x-, y- or z-coordinates are used depending on the settings within the traveltimes radiogroupbox (x-project, y-project or z-project). This setting is useful for data which have been acquired along one coordinate axis (e.g. refraction data). If calculate travelttime differences using all coordinates has been chosen all coordinates are used. This setting is useful for data which have been acquired along different coordinate axis (e.g. transmission data).

The total absolute timedifference, the total timedifference, the RMS deviation and the number of identical positions are displayed. The total absolute timedifference defines the sum of the absolute timedifferences independently from the sign (pos.

or neg. difference). This value gives you an estimate of the overall adaptation. The total timedifference defines the sum of the timedifferences taking into account the sign of the differences. This value gives you an estimate whether the mean model leads to too small or too big traveltimes, e.g. an interface is too high or too deep. The RMS deviation defines the Root Mean Square deviation. The number of identical positions is shown together with the total number of data picks in order to have an estimate over the significance of the analysis.

The option **color ray for dt's** allows to colour the rays based on the actual traveltimedifferences. Three colours are used: green is used if the difference is smaller than the total RMS deviation, red is used for larger and blue for smaller calculated traveltimes. A factor (in % of the RMS deviation) can be entered in order to change the crossover from green to red/blue. The picture below clearly indicates too small traveltimes on the left hand side obviously due to a too low layer boundary between 0 and 20 m.



The option **generate pickfile (2-way traveltimes)** allows to generate a pickfile consisting of the 2-way reflection traveltimes generated from the layer boundaries of the current model (except the uppermost layer boundary). The vertical gradients are taken into account. The 2-way reflection traveltimes are calculated up to the uppermost layer (corresponds to the model top in case of no topography). The number of the layer boundary is stored within the pick code. Example of application: conversion of a model taken from refraction data into 2-way traveltime data for a comparison with a seismic reflection section.


The option **generate pickfile (depths)** allows to generate a pickfile consisting of the depths of the layer boundaries of the current model (except the uppermost layer boundary). The number of the layer boundary is stored within the pick code. Example of application: conversion of a model taken from refraction data into 2-way traveltime data for a comparison with a seismic depth reflection section.

Help Menulitem: This item activates the main help menu or displays some information about memory and disk usage etc..

Exit Menulitem: Leave the module modelling.

The option **fill** allows to fill the model layer with colors automatically assigned to one physical parameter of each model point. You may determine within the model generation input of model parameters menu (chap. 4.4.3) which physical parameter is used. Click on the wanted parameter in the upper right corner table. By default the first parameter is set (marked by a x). The option is also valid for the print output. When filling the model a Reflexw rasterfile will be automatically created. By default a dummyfile will be used. If you enter a name under **rasterfilename** the entered filename will be used for the Reflexw rasterfile. The file will be stored under rohdata under the current project directory.

The **traveltime** radiogroup within the second lower panel controls how the traveltimes (calculated or loaded) are displayed. With **x-project** activated the traveltimes are plotted over the x-distance axis. With **y-project** activated the traveltimes are plotted over the y-distance axis. With **z-project** activated the traveltimes are plotted over the z-depth axis.

The option **highlighted shot** may be used for choosing a shot number to be highlighted. The traveltimes and the rays which correspond to this shot number will be highlighted. With the option **hide other shots** activated all the shots except the highlighted one will not be displayed. The speedbutton  resets the highlighted shot to 0.

The **sec.coord.** radiogroup within the second lower panel controls the choose of second coordinate e.g. for the raytracing and the 2D-tomography. The first coordinate is always x. With **y-coord.** activated it is assumed that the y-coordinate represents the second coordinate axis. With **z-coord.** activated it is assumed that the z-coordinate represents the second coordinate axis. The sec.coord. option is used for example when calculating traveltimes using the raytracing or when performing a 2-D tomography with 3D-data. If you are not sure

with this option please use the option check rays or show rays within the tomography panel in order to check the geometry. The last chosen option is saved when leaving the program.

The speedbutton **V** or **velocity adaptation** under analyse: allows to determine a velocity by creating a straight line with two mouse clicks within the traveltime window. With this button activated the program waits for the input of two datapoints for generating a straight line of distinct velocity. Pressing the left mouse button determines the first datapoint and by moving the mouse with pressed button the line with the current velocity is shown. Leaving the mouse button holds the current line.

4.1 Modelling File MenuItem

From this menu you may choose your project directory, load, save, reset and print an existing model and copy the current model to the clipboard.

ProjectDir: Selection of a new project. After selecting this option the directories included in the current project directory are listed. From these you can choose the project you want to work with.

load model: Use this option to load an existing model. Models which are built using the DOS programs REFLEX and REFRA can also be loaded. The data should be stored in the directory MODEL (default model directory) of the current project directory but it is also possible to load the data from any other directory or drive.

save model: Saves the current model in the new format used by REFLEXW. The model is stored automatically in the directory MODEL of the current project directory and has the extension MOD. Activating the option **old format** (max. number of layer points 100) allows to save the model using the old format (for Reflexw version up to 7.5).

reset model: Reset all model parameters. All model parameters apart from the dimensions are reset.

load from ASCII: allows to import the coordinates and the parameters of the layer boundaries from an ASCII-file. The format of the ASCII-file must be the same as used with the option export to ASCII. Each line of the ASCII-file contains all informations about one point of a layer boundary:

electromagnetic: layer-nr. X-coord. Z-coord. ϵ μ σ $d\epsilon/dz$ $d\mu/dz$ $d\sigma/dz$

seismics: layer-nr. X-coord. Z-coord. Vp Vs ρ dVp/dz dVs/dz $d\rho/dz$

The current wave type as well as other options like topography and the FD-parameters are not changed. Only the model borders (min./max x- and z-coordinates) are changed if the current model borders are smaller than the min./max. coordinates of the layer boundaries within the ASCII-file.

export to ASCII: allows to export the coordinates and the parameters of the layer boundaries into an ASCII-file. Two formats are supported:

each model point: The file will be stored under the path ASCII under the projectdirectory using the extension MOD. Each line of the ASCII-file contains all informations about one point of a layer boundary:

electromagnetic: layer-nr. X-coord. Z-coord. ϵ μ σ $d\epsilon/dz$ $d\mu/dz$ $d\sigma/dz$

seismics: layer-nr. X-coord. Z-coord. Vp Vs ρ dVp/dz dVs/dz $d\rho/dz$

csv tabella: allows to export the actual model into a CSV tabella with each line containing position, depth, velocity and optionally altitude of each layer (see example below for a 3 layer model with activated altitutde, dimension in ft). The increment of the positions may be freely entered. The naming is automatic (modelfilename plus extension csv, stored under the subdirectory ASCII).

position	depth surface	altitude surface	velocity layer 1	depth layer 1	altitude layer 1	velocity layer 2	depth layer 2	altitude layer 2	velocity layer 3
0.00	0.00	522.50	1122.87	6.42	516.08	4104.55	28.63	493.87	14000.00
6.00	0.00	521.95	1131.51	6.32	515.62	4104.55	28.63	493.31	14000.00
12.00	0.00	521.48	1140.15	6.34	515.13	4104.55	28.76	492.72	14000.00
18.00	0.00	520.86	1148.79	6.60	514.26	4104.55	28.95	491.91	14000.00
24.00	0.00	520.28	1157.43	6.87	513.41	4104.55	28.82	491.47	14000.00
30.00	0.00	519.67	1166.07	6.98	512.69	4104.55	28.30	491.37	14000.00
36.00	0.00	519.02	1163.29	6.88	512.15	4104.55	28.12	490.90	14000.00
42.00	0.00	518.33	1152.37	6.60	511.73	4104.55	28.19	490.14	14000.00
48.00	0.00	517.71	1148.01	6.56	511.15	4104.55	28.72	488.99	14000.00
54.00	0.00	517.22	1152.84	6.47	510.74	4104.55	29.69	487.53	14000.00
60.00	0.00	516.68	1157.66	6.47	510.21	4104.55	30.65	486.04	14000.00
66.00	0.00	516.19	1162.49	6.18	510.01	4104.55	31.13	485.06	14000.00
72.00	0.00	515.56	1165.41	5.96	509.60	4104.55	31.12	484.45	14000.00
78.00	0.00	515.10	1147.40	6.04	509.06	4104.55	30.91	484.19	14000.00

load additional model: activating this option allows to load and plot additional models. The additional models cannot be edited and are only plotted for comparison.

load 1D-models: activating this option allows to load several 1D-models which have been generated within the CMP velocity analysis (chap. 2) or within the 2D-dataanalysis when using the intercepttime method within the velocity adaptation (chap. 1.12.1) panel. The layer boundaries of the individual 1D-models are marked by colored signs. A two-dimensional model may be composed from the one-dimensional models using the option create 2D-model within the modelling Edit MenuItem (chap. 4.2).

print model: Use this option if you want to print out the model.

copy to clipboard: Use this option if you want to copy the current model to the clipboard.

load data traveltimes: allows to load picked data traveltimes. The plot color of the traveltimes is taken from the current set symbol font color. It is possible to load REFLEX formatted PCK files as well as 2D-columns ASCII with each line containing *distance* *traveltime*

ASCII-3D-tomography with each line containing (see also chap. 1.12.2.1)

traveltime code x-transmitter y-transm. z-transm. x-receiver y-receiver z-receiver

ASCII-2D-tomography with each line containing (see also chap. 1.12.2.1)

traveltime code x-transmitter y-transmitter x-receiver y-receiver

In addition you may load ASCII-matrix data (option Shot-Rec.Matrix, m shots and n receivers) which has the following format:

1. line contains the x-positions of all m shots
2. line contains the z-positions of all m shots
- 3.-n.lines:
 - 1.column: x-receiver position
 2. column: z-receiver position
 - 3.-m. Column: picked traveltimes

example:

```

shot position -100.00 -50.00 3.75 41.25 78.75 132.50 182.50
GP pos. Elev. 0.00 0.00 0.00 0.00 0.00 0.00 0.00
0.0 0.00 110.00 59.00 10.00 57.00 78.00 140.00 175.00
7.5 0.00 115.00 62.00 10.50 50.00 73.00 137.50 170.00
15.0 0.00 120.00 65.00 20.00 46.00 69.00 132.50 168.00
22.5 0.00 124.00 71.00 28.00 42.50 62.50 130.00 165.00

```

30.0	0.00	128.00	78.00	39.50	25.00	60.00	130.00	165.00
37.5	0.00	130.00	94.00	47.00	11.00	50.00	122.50	162.50
45.0	0.00	135.00	105.00	56.00	10.50	45.00	120.00	160.00
52.5	0.00	138.00	116.00	60.00	22.00	37.50	115.00	158.00
60.0	0.00	141.00	119.00	62.50	30.00	31.50	109.00	154.00
67.5	0.00	143.00	121.00	68.00	38.00	24.00	102.50	147.00
75.0	0.00	147.00	126.00	72.00	45.00	15.00	95.00	141.00
82.5	0.00	150.00	130.00	76.00	50.00	9.50	85.00	135.00

If any of the ASCII-formats has been used the timedimension in case of seismic wave propagation will be queried (ms or μ s).

load calc. traveltimes: allows to load calculated traveltimes. The dataformat of the calculated traveltimes is the same PCK-REFLEX format like for the data traveltimes but the plotting colors are different. The color corresponds to the shot no stored with the traveltimes.

export calc.traveltimes to ASCII: allows to save the currently loaded calculated traveltimes into an ASCII-format. The format corresponds to the ASCII-3D-tomography format within the Pick save MenuItem (chap. 1.12.2.1). Each line contains the following values: `traveltime(10:xx)`, `code(10:0)`, `x-traceheadercoordinate of the transmitter`, `y-traceheadercoordinate of the transmitter(10:xx)`, `z-traceheadercoordinate of the transmitter(10:xx)`, `x-traceheadercoordinate of the receiver(10:xx)`, `y-traceheadercoordinate of the receiver(10:xx)`, `z-traceheadercoordinate of the receiver(10:xx)` where `xx` defines the number of decimal places given within the global settings menu (chap. 1.2.1). The file has the extension TOM and will be stored under the path ASCII under the current projectpath. The format corresponds to that format needed by the 2D and 3D-tomographic traveltime-inversion.

export data traveltimes to ASCII: allows to save the currently loaded data traveltimes into an ASCII-format. The format corresponds to the ASCII-3D-tomography format within the Pick save MenuItem (chap. 1.12.2.1). Each line contains the following values: `traveltime(10:xx)`, `code(10:0)`, `x-traceheadercoordinate of the transmitter`, `y-traceheadercoordinate of the transmitter(10:xx)`, `z-traceheadercoordinate of the transmitter(10:xx)`, `x-traceheadercoordinate of the receiver(10:xx)`, `y-traceheadercoordinate of the receiver(10:xx)`, `z-traceheadercoordinate of the receiver(10:xx)` where `xx` defines the number of decimal places given within the global settings menu (chap. 1.2.1). The file has the extension TOM and will be stored under the path ASCII under the current projectpath. The format corresponds to that format needed by the 2D and 3D-tomographic traveltime-inversion.

export Reflexw rasterfiles: allows the conversion of REFLEX-formatted data (e.g. the tomographic results or a rastered model (option fill)) into different export formats (e.g. ASCII-GRD) - see also Export Menu (chap. 1.6).

Exit: Leave the module modelling.

4.2 Modelling Edit MenuItem

Undo: The option allows to undo the last model change.

create 2D-model from 1D-models: This option allows the automatic combination of several 1D-models to a 2-dimension model. The 1D-models must be loaded first using the option load 1D-models under modelling file MenuItem (chap. 4.1). The number of layers within each 1D-model should be equal because the layers are combined corresponding to their index number. The program automatically controls this number. The depths of the layers are linearly interpolated. The parameter table with the velocity and the vertical velocity gradient are automatically updated. If only 1D-model has been loaded the layers will be extrapolated.

create several 1D-models: This option allows the generation of single 1D-models from the actual model at equidistant positions. The number of 1D-models will be asked for. The filenames are generated from the actual model filename plus ' ' and a running number starting at 1. The 1D-models are stored under the path ROHDATA and have the extension VEL. The option might be useful if a migration or stacking shall be done based on these 1D-models. For this purpose enter the CMP-velocity module and create a 2D-model from the 1D-models in the same matter as if the 1D-models would have been derived from single shots or CMP's.

Restrictions: a topography will not be taken into account and zmin should be set to 0.

Exchange x-y-coordinates from datatraveltimes: normally the program uses the x-shot and x-receiver coordinates for displaying the datatraveltimes. This option allows you to use the y-shot and y-receiver coordinates instead. First you have to load the datatraveltimes (option file/load datatraveltimes). Then use this option in order to display the data based on the y-shot and receiver coordinates. The option might be useful if you have acquired two perpendicular refraction profiles and you entered a y-geometry for one of the profile (remark: this is normally not recommended because all refraction interpretation tools are based on the x-coordinates).

Fill actual model: allows to fill the model layer with colors automatically assigned to one physical parameter of each model point (see also speed option fill). You may determine within the model generation input of model parameters menu (chap. 4.4.3) which physical parameter is used. Click on the wanted parameter in the upper right corner table. By default the first parameter is set (marked by a x). The option is also valid for the print output. When filling the model a Reflexw rasterfile will be automatically created. By default a dummyfile will be used. If you enter a name under **rasterfilename** the entered filename will be used for the Reflexw rasterfile. The file will be stored under rohdata under the current project directory.

mix rasterfile with with lower layer boundary: allows to combine an existing rasterfile with the lowermost layer boundary. A new Reflexw rasterfile with the filename specified under **rasterfilename** will be generated which contains the same values as the old rasterfile except at the rasterpoints below the lowermost

layer boundary where their physical parameters are used. The option might be useful for example if a lowermost reflection has been interpreted (option adapt reflector) based on a rastered overburden (e.g. from a tomographic inversion).

flip model in distance direction: activating this option flips the actual model in distance direction.

4.3 Modelling View MenuItem

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

interpolated physical parameters: activate this option if you want the physical parameters to be displayed in the status panel on the top (par:) when moving the mouse. Analogous to the option fill you may determine within the model generation input of model parameters menu (chap. 4.4.3) which physical parameter is used. Click on the wanted parameter in the upper right corner table. By default the first parameter is set (marked by a x).

modelpoints: Allows to display or to hide the model layer points.

modellayers: Allows to display or to hide the model layer boundaries.

modelparameters: With this option activated the physical parameters (one single set of parameters) are displayed below the defined model points. The program controls if identical physical parameters for subsequent layerpoints are present and only displays the middle one. Analogous to the option fill or interpolated physical parameters you may determine within the model generation "input of model parameters menu" (chap. 4.4.3) which physical parameter is used. Click on the wanted parameter in the upper right corner table. By default the first parameter is set (marked by a x).

data traveltimes: Allows to display or to hide the data traveltimes which have been loaded using the option load traveltimes under modelling file MenuItem (chap. 4.1).

calc. traveltimes: Allows to display or to hide the currently calculated traveltimes or the traveltimes which have been loaded using the option load calc. traveltimes under modelling file MenuItem (chap. 4.1).

calc. rays: Allows to display or to hide the current calculated rays.

show data traveltimelpoints: Displayed data traveltimes are marked with crosses if this option is activated.

show calc. traveltimelpoints: Displayed calculated traveltimes are marked with crosses if this option is activated.

show data traveltimelines: Displayed data traveltimes of a shot are connected with straight lines if this option is activated. The options use shot for discr., use code for discr. and max.diff. for traveltimelines within the global settings menu (chap. 1.2.1) control which traveltimelpoints will be connected.

show calc. traveltimelines: Displayed calculated traveltimes of a shot are connected with straight lines if this option is activated. The options use shot for discr., use code for discr. and max.diff. for traveltimelines within the global settings menu (chap. 1.2.1) control which traveltimelpoints will be connected.

show additional models: allows to display or to hide the loaded additional models (see also option load additional models under modelling file MenuItem (chap. 4.1)).

show 1D models: allows to display or to hide the loaded 1D models (see also option load 1D models under modelling file MenuItem (chap. 4.1)).

show additional rasterfile: allows to display any rasterfile in addition to the model. This rasterfile may be a REFLEX-formatted file or a FD-rasterfile (extension FDP, standard outputpath ASCII) generated during the rastering of the FD-modelling, the raytracing or the tomographic inversion. The option is also valid for the print output.

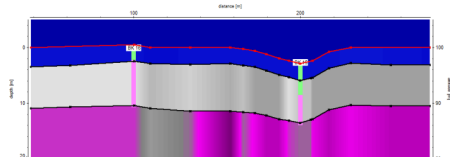
Core data/1D models: activate this option if you want to display core data (1D-depth distributions) as vertical bars onto model. Precondition is that either the option fill has been activated or any other additional rasterfile (e.g. a tomographic result) has been loaded. It is possible to select a choosable number of core datafiles. The core datafile should be stored under the path ASCII under the current project directory and must have the extension cor(e.g. c:\reflex\demodata\ascii\test.cor). A core datafile contains the informations of several cores associated with the current model. One core block within the file has the following format:

1. line: CORE + any identification
 2. line: current location (x,y and z) within the 2D-line (floating point format)
 3.-x. line: layernumber thickness quality-factor mean-velocity layer-velocity
 The dimension of the thickness and of the location is identical to the current distance dimension of the model. The velocities will be ignored.
 A block ends when the program finds the identifier CORE or at the end of the file.
 example:

```
CORE BK 16
      100  0   100.5000
  1    2.750000  1    2
  2    8.450000  1    2
CORE BK 15
      200  0   97.90000
  1    6.400000  1    2
  2    5.450000  1    2
```

The **x-location** corresponds to the distance. The **y-location** will not be used. The **z-location** will be considered. With the option altitude activated the z-locations are assumed to be elevations, if deactivated they are given in depths.

The max. number of different layers within one coreblock is 100 (identical to the max. number of layers within the layershow). The colors for the individual layerbars are identical to the colors within the LayerShow MenuItem. With the option **show quality** within the global settings activated a second layerbar is shown indicating the quality factor with the following color assignment: 1 - white, 2-yellow, 3-red. Within the global settings menu you may also change the width of the layerbars and activate or deactivate some checking parameters.



4.4 Model generation

The first paragraph of this chapter gives a short introduction of the steps which have to be done during the generation of a new model. The following paragraphs describe the various possibilities to change this 'raw' model in order to adapt it to the physical reality which shall be simulated. PLEASE NOTE: Independent of the wanted simulation (FD-calculation, Tomography or ray-tracing) the models are all generated in the same way.

4.4.1 Getting started

The first steps to generate a model are to define the model size (start and end coordinates in x- and z-direction [m]), the model file name with which the model should be stored and the wanted wave type during the simulation (electromagnetic, seismic(elastic), acoustic and sh-seismic). PLEASE NOTE: The model file name should not exceed 6 characters (test10, e. g.) to ensure a correct generation of the output file name of the calculated data (see below). All these parameters can be set in the first lower panel.

After clicking the reset button in the first lower panel the model and the first interface (layer nr. 1) are displayed. This first layer interface is automatically generated and represents the upper boundary of the model (i.e. the 'raw' model always consists of one layer with one interface at its top).

The next step is to define the model parameters of the first layer by using the table in the input of model parameters window which displays all the model parameters belonging to the current layer. This is done in the same manner as for any other layer and will therefore be generally explained below.

After having done these steps, the generation of the 'raw' model is finished and the adaption of the model to the reality can start.

With the option altitude activated a second axis on the right is shown with the altitudes based on the entered reference level. The current altitude is calculated from: reference level - current z-value.

The option ycoord allows to enter the coordinate value into the third (y) direction. This value is used for the generation of the 3D-model from 2D-cuts for the 3D-FD calculation (only the simulation of the electromagnetic wave propagation, see also chap. 4.5.4).

4.4.2 Generating a new layer

By clicking on the new button the layer nr. will automatically be adapted to (n+1), whereby n is the nr. of existing layer interfaces. (Alternatively you can count up to the maximum possible layer nr.) If the radio buttons in the first lower panel are set to layer and set point all needed points of the new layer interface can be set inside the model by means of the left mouse button. The interface is interpolated linearly between the layer points. Another possibility to set the layer points is to directly fill in their coordinates into the table in the input of model parameters window (see following paragraph).

After having set all points, please click on the button extrapolate or hor.extrap. in the second lower panel to extrapolate the layer to the model boundaries, the

nearest layer, respectively. 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, rectangle, triangle and lens (see below). 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).

The physical parameters of the layer (ϵ , μ and σ , V_p , V_s and ρ , respectively) can only be entered using the table in the input of model parameters window (see following paragraph).

PREDEFINED SYMBOLS: If the radio buttons in the first lower panel are set to circle, rectangle, triangle, lens, respectively, and set point, the location of the middle point, the focal point, respectively, can be set in the model by means of the left mouse button or by using the option place object within the input of model parameters window using the midpoint coordinates center x-pos. and center z-pos. PLEASE NOTE: It is not necessary to care about the fact that the predefined symbols exist of two interfaces. The layer nr. is adapted automatically and after the generation of the chosen object the two interfaces are treated independently. The needed physical and additional parameters like the radius of the circle or the extension of the rectangle, triangle and lens are entered in the input of model parameters window (see following paragraph).

4.4.3 Input of model parameters window

The input of model parameters window is used to set, display and change all the parameters belonging to the current layer. This window comes to the fore by means of the right mouse button if the cursor is situated inside the model.

In the table at the bottom of the group box all the points of the current layer are listed in a rising sequence according to their x-coordinates. In addition to the coordinates of the points their corresponding physical parameters are listed dependent on the chosen wave type and if the physical parameters should vary gradually with depth (EditGradients is chosen in the ControlBox).

All parameters can be set or changed by simply editing the corresponding fields of the table. By clicking the **update model** button in the ControlBox the model values and the model display are updated. (A new point is added at the end of the table. The points are automatically resorted during the update.)

Another possibility to set or change the parameter values of some points is to use the field situated in the upper right corner of the input of model parameters window. In this field all the necessary physical parameters (ϵ [], μ [] and σ [S/m] or V_p [m/s], V_s [m/s] and ρ [g/cm³], respectively) are listed and can be set individually. 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.

Clicking the **take over** button in the ControlBox leads to the updating of the parameters at the selected points. (For updating the model please use the update button in any case!). With the option take over all activating the action leads to the

updating of the parameters at all layer points. Activating the option EditGradients extends the table by the vertical gradients.

If the parameters in this field are set before the first point of a layer is set interactively, these parameters will be assigned to all the layer points.

The **import (x,z)** button in the ControlBox allows the import of (x,z)- or (x,y,z)-coordinates dependent on the setting of the file of type within the file dialog window.

In both cases the data must have been saved in ASCII format:

(x,z)-coordinate: one line contains the x-coordinate (=distance) and the z-coordinate which defines the depth.

(x,y,z)-coordinate: one line contains the x- and y-coordinates (e.g. UTM coordinates) and the z-coordinate which defines the depth. 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.

If specified the 3. (4.) and 4. (5.) column may contain the physical parameter Vp or epsilon and the corresponding vertical gradient, the 5. (6.) and 6. (7.) column Vs or mue and the corresponding vertical gradient and the 7. (8.) and 8. (9.) column rho and sigma and again the corresponding vertical gradient. If these parameters are not specified the following holds true: if no physical parameters already exist the current parameters are taken over. If the physical parameters of the current layer already exist those parameters which are most close to one xz-coordinate point are taken over for that point.

if the actual layer is 1 it is assumed that a topography will be introduced. The option topography will be automatically activated if deactivate and an interpolation in distance (x)-direction of the nearest existing parameters will be done instead of simply using the most nearest ones. If the option altitude is activated in addition the z-values within the ASCII-file define altitudes and not depths for layer 1. The reference level should be set at least to the maximum existing altitude value. The depth values are then calculated from:
reference level - altitude values.

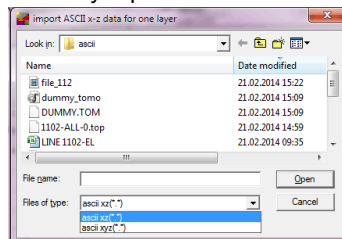
After having performed the option the model will be automatically updated.

The **export (x,z)** button allows the export of the xz-coordinates of the current layer. Two formats are supported:

model x-z data: format corresponds to the format also used for the option import (x,z). One line contains the x-coordinate and the z-coordinate (depth or altitude for the 1. layer with activated option altitude based on the entered reference level).

The following columns contain the physical parameters together with the corresponding vertical gradients (e.g. Vp, dVp/dz, Vs, dVs/dz, Rho, dRho/dz)

top xyz data: format corresponds to the topography fileformat used e.g. for the processing options topography migration and correct 3D-topography. One line contains the x-coordinate, the y-coordinate and the altitude (based on the entered reference level).



PREDEFINED SYMBOLS: If a circle, a rectangle, a triangle or a lens shall be integrated into the model two values for each parameter of the field situated in the upper right corner can be set: The upper parameters indicate the values of the upper interface (i.e. inside the object), the lower ones the values of the lower interface (i.e. below the object). Usually the parameters of the lower interface

(outside the object) are in accordance with the parameters in the layer in which the object has been embedded.

In case of a rectangle, triangle or lens the length (extension in x-direction [m]) and the width (extension in z-direction [m]) must be set. In addition for the rectangle, there is a possibility to set a slope in °, which allows to generate parallelograms with the upper and lower border having the chosen slope value.

If a circle shall be set, only the radius has to be chosen.

The option place object allows to set a new symbol object using the midpoint coordinates center x-pos. and center z-pos.

With the option **TransitionZone** activated the actual model boundary is assumed to be rather a transition zone than a sharp boundary. The size of the transition zone may be entered within the option size of transition zone. The transition zone is centered at the original position of the model boundary. Using the transition option might be useful for a subsequent tomography with a start model with boundaries resulting from a previous wavefront inversion.

The option **RandomLayer** within the model input menu allows to specify statistic parameters for a random perturbation of the physical parameters of the individual layer. It is possible to choose between fluctuations (see picture on the right - 1. layer) and discontinuous perturbations (see picture - 2. layer). Different spatial distributions as well as different statistic distributions for the physical parameters are available.

The correlation lengths in x- and z-direction define the average size of the inhomogeneities.

The variance defines the strength of the perturbation for the first physical parameter (either p-velocity or epsilon).

The parameters var. par. 2 and var. par. 3 define the strength for the 2 other parameters (s-velocity or μ and density or sigma) respectively.

The distribution functions are the same for all physical parameters.

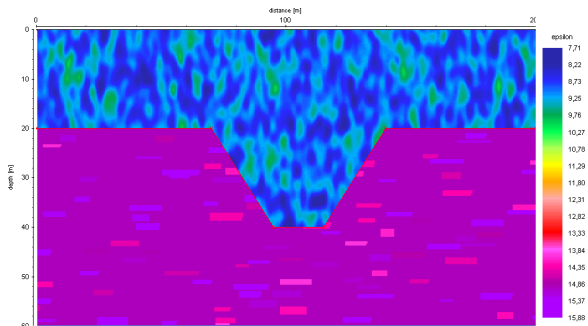
The parameter density(%) allows you to restrict the density of the perturbations (density must be greater than 0 and smaller/equal 100). The parameter does not exactly represent the resulting density of the perturbations especially for small correlation lengths and for the type discontinuous. Therefore it might be necessary to change this value by trial and error if the resulting density does not fit the wanted needs. With the type discontinuous the single perturbations do not overlap for densities smaller than/equal 75 %. For greater densities an overlapping is allowed.

The parameter mean value defines a static level for the perturbations which will be added to all perturbation values. It is possible to define a positive or negative mean value.

By deactivating the option random start the random function repetitively generates the same specific sequence of random numbers if the other parameters and the discretization interval do not change. Activating the option random start always generates a new sequence of random numbers.

As the random distribution depends on the used raster increment the results from the option fill and from the "normal" FD-rastering differ. Therefore if you want to check your FD-rasterfile you always should use the option "show additional rasterfile" or "Reflex rasterfile" instead of the option fill.

If an object has been placed within a random layer (e.g. a circle or a rectangle) you must define the same random parameters for the lower boundary of the



act. layer no.: 1		Random layer	
take over	update model	type: fluctuations	variance: 2
import (x,z)		spatial: normal	var. par.2: 0
		phys.par: white	var. par.3: 0
<input type="checkbox"/> take over all		corr.length(x): 5	density(%): 100
<input type="checkbox"/> EditGradients		corr.length(z): 5	mean value: 0
<input checked="" type="checkbox"/> RandomLayer		corr.length(y): 0	<input type="checkbox"/> random start

act. layer no.: 2		Random layer	
take over	update model	type: discontinuous	variance: 2
import (x,z)		spatial: white	var. par.2: 0
		phys.par: white	var. par.3: 0,1
<input type="checkbox"/> take over all		corr.length(x): 20	density(%): 20
<input type="checkbox"/> EditGradients		corr.length(z): 1	mean value: 0
<input checked="" type="checkbox"/> RandomLayer		corr.length(y): 0	<input type="checkbox"/> random start

integrated object and the option random start should be deactivated for both this lower boundary and the boundary of the random layer.

It is also possible to generate a **3D-random datafile**. If the option 3D is activated either within the ray (3D-ray) or within the FD-input box (3D-FD) the corr.length (y) in y-direction must be entered in addition. The type discontinuous is not available for a 3D-datafile. Precondition for the application of the random option for a 3D-file are:

- the random parameters for all layers are taken from the 2D-modelcut with the smallest y-coordinate
- the random parameters must be the same for all 2D-modelcuts
- all layers must exist within each used 2D-modelcut
- changes of the random parameters must be saved first before rastering because the individual 2D-modelcuts are read from the modelfiles.

4.4.4 Changing an existing layer

All changes but the removing of a layer can be done using the input of model parameters window. In addition, the position/form of an existing layer can be changed interactively by using different options in the first, second lower panel, respectively.

PLEASE NOTE: The radio button layer in the first lower panel has to be activated to carry out any interactive changes on the current layer. Then, e.g. the layer, which should be changed, can be chosen interactively by means of the left mouse button while choose layer is activated and the cursor is situated near the layer interface. (It is also possible to choose a layer by adjusting the corresponding layer nr. in the first lower panel.) Any point of the current layer can be changed or removed interactively (change p. or remove p.). New points can also be set (set point). For changing a layer point, please choose the desired point by means of the left mouse button, hold it until the desired locus is reached and then release the left mouse button.

With edit layer the radio groupbox edit current layer is activated and some edit functions which influence not only one layer point but the whole current layer can be carried out. The corresponding parameters act on the actual layer nr. if the button apply will be clicked or if any layer boundary will be chosen using the left mouse button.

edit current layer			tot. ver.move	amount	extrapolate	add topog.
<input type="radio"/> remove	<input type="radio"/> vert.move	<input type="radio"/> randomize	tot. hor.move	8	hor.extrap.	remove top.
<input type="radio"/> average	<input type="radio"/> hor.move	<input type="radio"/> polynomial	points	100	lin.extrap.	sort->depths
<input checked="" type="radio"/> run.mean	<input type="radio"/> comb.f.ad.					

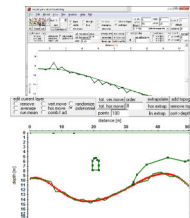
The layer can be removed (remove), vertically or horizontally moved (vert. move or hor. move) by a certain amount [m], which has to be entered in the amount box. In addition, the topography of the current layer boundary can be averaged by a given number of layer points, whereby the number of layer points has to be entered in the amount box. The option running mean allows to average the chosen layer using a running mean over a choosable number of points option (amount).

The option comb.f.add. allows to combine the actual layer with the same layer of the loaded additional model. The option might be useful for example in order to combine two different layer parts.

The option repl.by.ad. allows to replace the actual layer with the same layer of the loaded additional model.

The option randomize allows to generate a rough layer surface. The parameter xsize defines the correlation length in x-direction - the parameter zsize in z-direction.

The option polynomial allows an adaptation of any existing layer by a polynomial of given order. The order as well as the number of points for the resulting layer may be defined.



The option tot. ver.move allows to move all layers vertically by the entered amount.

The option tot. hor.move allows to move all layers horizontally by the entered amount.

The option combine layers allows you to combine a different number of layers to one single layer. After having activated the option the layer nr. of the layers to be combined are queried. If the total number of layer points exceed the max. number of layer points (250) an averaging is automatically done. Use this option for example if you have inverted one single layer within different wavefront inversion steps using different layer numbers.

The option extrapolate allows the automatic interpolation between two interfaces as well as the extrapolation of the interface at the border. 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). This is necessary for the following FD-rastering.

The option hor.extrap. allows the extrapolation of the interface only to the left and right borders of the model using a horizontal line. Use this option instead of extrapolate if you want the interface to be only extrapolated to the vertical borders. The option lin.extrap. allows the extrapolation of the interface only to the left and right borders of the model by using the slope of the first/last model points. Use this option instead of extrapolate if you want the interface to be only extrapolated to the vertical borders.

The option add topog. adds the topographic interface (layer 1) to all existing layers.

The option remove top. removes the topographic interface (layer 1) from all existing layers.

The option sort->depths allows to sort the layernumbers after depth. Activating this option might be useful if the layer numbering is not ascending and the option export to csv tabella is wanted. The sorting after depths also includes the first layer if topography has been activated.

The last change can be undone using the undo option under the edit file menu item.

4.4.5 Topography

The position of the points of the first layer (layer nr. 1) can normally not be varied in depth (z-direction). To take into account a topography at the top of the model, i.e. to vary the layer points in depth, the option topography in the second lower panel has to be activated.

The option import (xz) within the input of model parameters menu can also be used in order to define a topography. For that purpose the option topography must be activated. If the option altitude is activated in addition the z-values within the ASCII-file define altitudes and not depths for layer 1. The reference level should be set at least to the maximum existing altitude value. The depth values are then calculated from:

reference level - altitude values.

If the first layer has a topography, this topography can be added or removed from ALL other layers by clicking ONE time the button add topog., remove top., respectively. PLEASE NOTE: Clicking several times adds or removes the topography several times!

If you are calculating synthetic traveltimes (see also raytracing 2D/wavefront inversion 2D) or FD-seismograms/radargrams (see also FD-model simulation) or if you are performing a tomography (see tomography) also all source and receiver positions above the topography are automatically placed directly below the topography. Therefore it is not necessary to redefine the z-positions of the source and the receivers if a topography will be used in addition.

4.5 FD-Model simulation

The FiniteDifference simulation technique is described in the following paragraphs.

4.5.1 FD-computation

The parameters necessary for the FD-computation can be entered if the **FDGroupBox** is activated (e.g., use the speed button **FD** in the first lower panel). The rastering of the model based on the current parameters and the FD-computation must be started from this box, too.

According to the given wave type the simulation on the basis of the Maxwell-equations or of the elastic, acoustic, SH respectively, wave equation takes place. The result of this calculation is the wave field depending on x, z and t (time). An explicit method is used, i.e. the wave field is calculated for a certain point of time from the preceding point of time. All the FD-parameters listed below are stored in the ASCII-file with the name Filename+the extension 'FDR' (test10.fdr, e. g.).

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 (see above). 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. For all wavelets except the Klauder wavelet the entered frequency is used for determining the min. wave length, for the Klauder wavelet the entered max. frequency is used. This may lead under certain circumstances to a too low value and you may try a slightly higher value of the space-increment. The max. time-increment depends on the max. velocity as well as on the given space-increment (approximately: $\Delta t \leq \Delta x / (2V_{\max})$). The critical time-increment is shown in the calc. critical values box as well. In addition, the critical time-increment of the current set space-increment is displayed there.

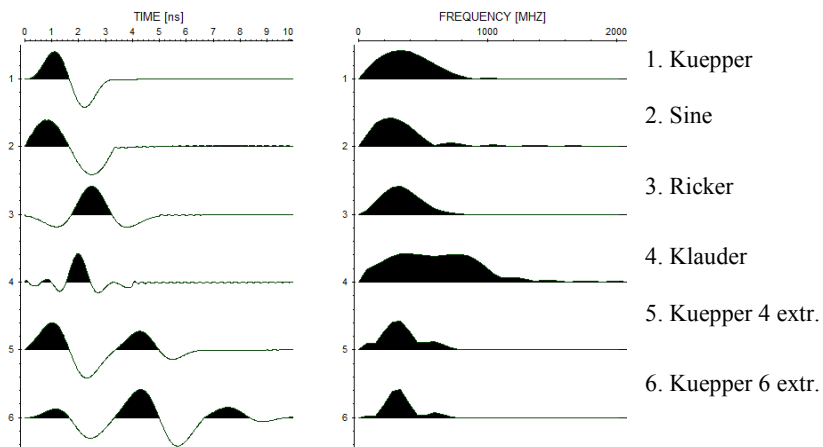
The input frequency shows the main frequency of the signal for the simulation in Hz (seismic), MHz (el.magnetic), respectively. TMax specifies the point of time in msec, nsec, respectively at which the simulation shall be terminated. The **boundary conditions** selection box enables you to choose one of the predefined model boundaries. In the case of electromagnetic wave propagation the model boundaries can be extended by an absorbing edge area (see chapter 4.5.1.2). In the case of seismic propagation an absorbing boundary condition is set on (see for example: Reynolds, boundary conditions for the numerical solution of wave propagation problems, geophysics, 43, 1099-1110, 1978). You may choose between the threedifferent boundary conditions A1, A2 and REY for the elastic wave propagation and REY for the acoustic one. The absorbing boundary conditions A1 and A2 do not show the problems with the instability at the model borders when a discontinuity is present but it is a little bit less efficient. Choosing free surface/borders means a free surface at all borders for the seismic wave propagation. Choosing reflecting means total reflection from the boundaries both for the electromagnetic and the seismic wave propagation (rigid borders).

The **source type** selection box allows the choice of a plane wave, a point source and an exploding-reflector model (not in the case of elastic wave propagation). According to the choice of the source the depth position (source z for plane wave and point source) as well as the lateral position (source x for point source) has to be given.

The **signal type** allows to choose any of the following different signal types: Kuepper, sine, Ricker, digitized signal, Kuepper 4 Extr. damped, and Kuepper 6 Extr. damped. In each case the parameter frequency defines the nominal frequency of the signal. The lower picture shows the different signal type together with its amplitude spectrum for a nominal frequency of 300 MHz (except Klauder wavelet - see below).

The digitized signal is taken from the first trace of a Reflexw formatted datafile. A possibly existing starttime within the Reflexw file will not be considered. The critical values for DeltaX and DeltaT are still be calculated from the manually entered frequency which should be the dominant frequency of the digitized signal. The Kuepper 4 and 6 Extr. damped types may be used in order to simulate a longer reverberating signal. The first 2 extrema remain undamped and then a cos-taper is used in order to damp the following 2 or 4 extrema.

The Klauder wavelet is the autocorrelation wavelet of a linear sweep signal. In this case the parameters min. and max. frequency define the frequency range of the original sweep signal (input in the same dimension like for frequency). The duration of the original sweep signal is fixed to the max. time range. The length of the Klauder wavelet is determined from the parameter frequency (signallength = $1/\text{frequency}$). The lower picture shows an example with a frequency of 250 MHz resulting in a signallength of 4 ns and a frequency range between 100 and 1000 MHz. To be considered: The Klauder wavelet is not minimum phase. Therefore it might be useful to adjust the time zero after the simulation.



According to the chosen wave type the excited fields, field components, respectively, of the source have to be set:

electromagnetic: setting of the excited field components Ex, Ey, Ez, Hx, Hy and H_z;

seismic(elastic): setting of p-source and s-source;

acoustic and sh-seismic: no selecting options because of a potential.

Default settings are Ey for the electromagnetic and p-source for the seismic(elastic).

The components of the calculated fields, which shall be registered and stored on disk can also be chosen dependent on the current wave type:

electromagnetic: setting of Ex, Ey, Ez, Hx, Hy and H_z;

seismic(elastic): setting of vert. displacement (WZ) and hor. displacement (UX);

acoustic and sh-seismic: no selecting options because of a potential (PT).

Default setting is Ey for the electromagnetic.

The option **free surface** in the case of seismic or acoustic wave propagation allows you to set the upper boundary always as a free surface even in the case of chosen absorbing boundary conditions.

The output file name for the computed data consists of the model file name plus an extension (according to the field component: EX, EY, EZ, HX, HY, HZ for the electromagnetic, WZ, UX for the seismic(elastic) or PT for the acoustic and sh-seismic calculation, respectively) and the successive number plus extension „DAT“. The data are stored in the directory ROHDATA of the current project directory.

The **output type** allows to set the wanted type of data output.

total wavefield: The complete wavefield is stored in 3D-REFLEXW format (test10ey.dat, e.g.) and can be viewed within the 3D-data-interpretation module (processing within the 2D-data-analysis module like any 2D-line).

single line: A single line is stored in REFLEXW format (test10ey.dat, e.g.) and can be viewed and processed within the 2D-data-analysis module.

several single lines: This option allows to compute a sequence of different modellings based on the current model. For further information see chapter 4.5.2. Choosing single line you have to specify the coordinates of the receiver line (recxstart, recxend, reczstart, reczend).

constant offset: The option is only available for the source type “point source”. The option allows to simulate a constant offset section for single point sources within one step. The **source-rec.offset** must be entered. The **rec. increment** together with the **rec start** and **end coordinates** define the geometry of the constant offset section. It is not possible to define a diagonal line. If the xend-xstart is greater than zend-zstart the line always lies horizontally at zstart. If xend-xstart is greater than zend-zstart the line always lies vertically at xstart. Example: In total 201 ZeroOffset (source-rec.off was set to 0) will be calculated from 0 to 50 m at the surface at z = 0,.

CMP: The option is only available for the source type “point source”. The option allows to simulate a CMP section for single point sources within one step. The **CMP-position** must be entered. The **rec. increment**

output type:	
ConstantOffset	
source-rec.off	0
rec. increment	0.25
output type:	
CMP	
CMP-position	20
rec. increment	0.25
rec x start:	22
rec x end:	30
rec z start:	0
rec z end:	0

together with the **rec start** and **end coordinates** define the geometry of the CMP section. The source coordinates will be automatically placed into opposite direction of the receiver coordinates.

Example: In total 33 shot/receiver combinations will be calculated with the CMP located at 20 m and the first shot at 18 m and the first receiver at 22 m. The last shot is located at 10 m and the last receiver at 30 m. Analogous to the constant offset outputtype it is not possible to define a diagonal line.

The given of x,z-scaling as well as time scaling controls, together with DeltaX and DeltaT, the sampling in x-,z- and time-direction for the storage of the wave fields on disc (e.g. x,z-scaling = 2 and time scaling = 4 means a sampling interval of $2 \cdot \Delta X$ and $4 \cdot \Delta T$). x,z-scaling and time scaling should be set as large as possible because in the case of the output type total wavefield the output usually needs a huge amount of disc space. In the case of the total wavefield output the program automatically controls the scaling parameters in such a way that the resulting number of points in all directions (x, z and time) does not exceed the max. number of datapoints for the 3D-datainterpretation (see chap. 3).

If you have entered a **topography** all source and receiver positions above the topography are automatically placed directly below the topography (see also model generation - topography, chap. 4.4.6).

Within the **3D-FD group** it is possible to activate the 3D-FD-calculation for the electromagnetic wave propagation - for further information see chap. 4.5.4).

DoRaster: The FD-program needs discrete 2-dimensional arrays of the physical parameters. The option DoRaster allows the rastering of the model parameters based on the current raster increment DeltaX. The max. number of raster points in z-direction is restricted to 32000.

During the rastering the model is filled with different colors. These colors do not correspond to any physical model parameter but indicate which areas of the model are attached to which model layers. PLEASE NOTE: If a FD-computation shall be done using a new model the first time, the rastering should be done at least one time with DoRaster. So the attachment of the model areas to the corresponding layers can be controlled and any error caused by the rastering can be avoided.

The 2D-arrays of the individual model parameters are stored under the path model under the current projectdirectory on different files with the following filenames: model file name + special extension. The different extensions are:

electromagnetic wave propagation: ϵ - FDP, μ - FDS, σ - FDD;

elastic wave propagation: Vp - FDP, Vs - FDP, ρ - FDD

acoustic wave propagation: Vp - FDP, ρ - FDD

sh-wave propagation: Vs - FDS, ρ - FDD

ExplodingReflector Model: source strength - FDQ, phase - FDI

Storing the arrays is done in an ASCII-format using the following block notation (all z-values for one x-point):

FDP and FDS-files: 10F12:4

FDD-files: 10F12:8

FDQ and FDI-files: 10F12:5

At the end of each block the line must be ended.

example - 15 values in z-direction, 2 blocks = 2 successive x-values:

```
10.00 10.00 10.00 10.00 10.00 10.00 10.00 10.00 12.00 12.00
```

```

12.00 12.00 12.00 12.00 12.00
10.00 10.00 10.00 10.00 10.00 10.00 10.00 10.00 12.00 12.00
12.00 12.00 12.00 12.00 12.00

```

If the rastering of the model parameters is not done using the option **Raster** of **REFLEX** please consider the following: the file with the extension **FDR** contains all the necessary parameters like signal length, max. time and the point number **Nx** and **Nz** in **x**- and **z**-direction, respectively. These numbers (+1 because counting starts at 0) must be identical to the number of points in **x**- and **z**-direction used in the raster files.

A message appears if undefined parameter values are found for ϵ und μ (electromagnetic) or V_p , V_s and ρ (seismic). The following default values are set in this case: ϵ : 1, μ : 1, σ : 0 S/m; V_p : 1000 m/s, V_s : 500 m/s, ρ : 1 g/cm³.

reflex rasterfile: if activated a **Reflexw** formatted rasterfile using the current raster parameters will be generated in addition when using the option **DoRaster** or **import**. The filename is taken from the **modelfilename** and the rasterfile is stored under **rohdata** under the current project directory. The file may be viewed under the **2D-dataanalysis**. If the option **3D** is activated a **3D-Reflexw** rasterfile is generated which may be viewed within the **3D-datainterpretation**.

export: the option allows to store the **FD-rasterfiles** onto one single **ASCII-file** with each line containing the following informations:

x-pos. y-pos. z-pos. $\epsilon(V_p)$ $\mu(V_s)$ sigma(density) sourcestrength sourcephase

The first line contains the following informations:

xpoints zpoints ypoints DeltaX

The number of places and of decimal places are given within the global settings menu (chap. 1.2.1) within the pick parameter panel.

import: the option allows to read the **FD-rasterfiles** from an **ASCII-file** with each line containing the following informations:

x-pos. y-pos. z-pos. $\epsilon(V_p)$ $\mu(V_s)$ sigma(density) sourcestrength sourcephase

The first line contains the following informations:

xpoints zpoints ypoints DeltaX

An error message appears if the current **FD-parameters** and the parameters stored within the **ASCII-file** do not match. Deactivating the option **raster** allows you to start a **FD-simulation** using the parameters stored within such an **ASCII-file**.

The options **export** and **import** may be used for an easier manual change of the rasterized physical parameters.

StartFD: The option allows the start of the **FD-program** **fdemseis.exe** or **fdemseis_64bit.exe** respectively if the option **64Bit_mode** has been activated. The **64Bit_mode** can only be used if a 64 bit Windows system is present. Then there is nearly no restriction concerning the maximum memory and the model size can be enlarged accordingly.

With the option **raster** activated first the chosen model will be rastered based on the current parameters and then the **FD-computation** will be started.

With the option **raster** deactivated (not possible with activated **3D-FD** option - see chap. 4.5.4) the computation is started immediately using the current raster values stored in the different rasterfiles (see option **DoRaster**). The program automatically controls if the size of the rasterfiles matches the currently set values of the raster increment and the model size. If not an error message appears. The

number of points in z-direction and x-direction must be: $(z\text{-size} / \Delta x) + 1$ and $(x\text{-size} / \Delta x) + 1$.

The program FDEMSEIS.EXE will normally be executed in the background, so that it is possible to go on with the work in REFLEXW. If the option background is deselected, the computation is faster but there is no possibility to work with REFLEXW until the FD-computation is finished.

BatchStartFD: batch start for different models. After having activated the option the model filenames for the FD-calculation are queried. Before using this option you have to save your models with the wanted FD-parameters.

4.5.2 Computation of several single lines

If the output type is set to several single lines, a sequence of different modellings based on the current model can be computed. Up to 999 different single shots may be simulated in one step. The source must be a point source.

An ASCII file FDSHOTS.* (FDSHOTS.1, e.g.) under the path MODEL must exist which contains in each line for a single modelling the shot, receiver positions, respectively, and the name of the file, in which the data shall be stored:

```
sourcepos sourcezpos recxstart recxend reczstart reczend modeloutputname
```

Again, the modeloutputname should not exceed 6 characters to ensure a correct generation of the output file name of the calculated data.

The number of lines in the file controls the number of different calculations.

example:

```
0.0 0.0 0.0 4.0 0.0 0.0 test10
1.0 0.0 0.0 4.0 1.0 0.0 test20
2.0 0.0 0.0 4.0 2.0 0.0 test30
3.0 0.0 0.0 4.0 3.0 0.0 test40
4.0 0.0 0.0 4.0 4.0 0.0 test50
```

Five simulations with different source and receiver positions will be generated. The resulting five single lines will be stored with different names in REFLEXW format in the directory ROHDATA of the current project directory (test10ey.dat, test20ey.dat, test30ey.dat, test40ey.dat, test50ey.dat, respectively, if the field component EY shall be stored and the wave type is chosen to electromagnetic).

After starting the computation (StartFD button) the ASCII file FDSHOTS.* has to be chosen whereupon the included shot geometry is displayed. If the shot geometry is correct the computation can be started, otherwise aborted.

The option **create several lines file** allows to generate such an ASCII-file. The parameters places and decimal places within the global settings menu are used for the format of the ASCII-file.

Activating **fixed line** allows you to define the geometry for a fixed receiver line for different shot points. The right panel shows an example of shots located from 0 (source z start) to 100 m (source z end) with a spacing (source incr.) of 5 m within a vertical borehole at 0 m (source x start and source x end). The receivers are located from 0 (rec. z start) to 100 m (rec. z end) within a second vertical borehole at 60 m (rec. x start and rec. x end).

Activating **moving line** allows you to define the geometry for a receiver line moving with the shots. The receiver start and endcoordinates will be updated for each shot using the entered source incr. The input parameters are the same as for the fixed line.

create SeveralLines file

☒ fixed line
☐ moving line

source x start: 0
source x end: 0
source z start: 0
source z end: 100
source incr.: 5
rec x start: 60
rec x end: 60
rec z start: 0
rec z end: 100
filename: ver
file extension: 0
generate close

4.5.3 Absorbing Range for the e.-m. wave propagation

For the electromagnetic FD-wavepropagation an absorbing range is used to decrease the disturbing reflections from the model boundaries. You have the possibilities between lin.absorbing-range, exp.absorbing-range, big lin.absorb.range and big exp.absorb.range.

In all cases it is assumed that the conductivity inside the absorbing range is continuously increased to a max. value starting from the current one. Entering lin.absorbing-range, big lin.absorb.range, respectively, a linear increase is used, entering exp.absorbing-range, big exp.absorb.range, respectively, an exponential increase is used.

The max. value of the conductivity σ_{end} is defined as:

$$\sigma_{end} = fac \cdot \omega \cdot \epsilon_{act} \cdot \epsilon_0$$

ϵ_{act} – actual ϵ of the last model point before the boundary

$$fac = \frac{Size}{50} \text{ for linear increase}$$

$$fac = \frac{Size}{20} \text{ for exponential increase}$$

$Size = 50$ gridpoints for small range

$Size = 100$ gridpoints for big range

With k - grid variable inside the boundary range the conductivity σ_k for the linear increase is defined as:

$$\sigma_k = \sigma_{act} + \frac{\sigma_{end} - \sigma_{act}}{Size} \cdot k$$

and for the exponential increase:

$$\sigma_k = \sigma_{act} \cdot \exp\left(\frac{k}{Size} \cdot \ln \frac{\sigma_{end}}{\sigma_{act}}\right)$$

σ_{act} – actual σ of the last model point before the boundary

4.5.4 FD-modelling 3D

Within the 3D-FD group it is possible to activate the 3D-FD-calculation for the electromagnetic wave propagation.

With the option **3D-FD** activated a 3D-FD-calculation is performed.

The parameter **Ycoord** allows to enter the coordinate in y-direction. The source and the receiver line are located at that y-direction

The parameters **ymin** and **ymax** define the model range in y-direction. The rasterincrement DeltaX also holds true for the y-direction.

The 3D-model is built up from different 2D-cuts lying within the xz-plane. The parameter **ycoord** (see chap. 4.4.1) defines the location of each cut in the y-direction. The option **load** allows you to define the wanted 2D-model cuts.

4.6 Tomography

If the TomographyGroupBox is activated (e.g., use the speed button TOMO in the first lower panel), the parameters necessary for the Tomography can be entered. The rastering of the model based on the current parameters and the Tomography must be started from this box, too.

The simulation technique Tomography allows the automatic adaptation of synthetic travel time data to real data based on a tomographic algorithm. Only transmitted (crosshole tomography) or refracted data (refraction tomography) may be used. The algorithm is based on an iterative adaptation (SIRT-Simultaneous Iterative Reconstruction Technique). Starting from the current model synthetic travel times are calculated. Either straight or curved rays (only 2D) are considered. These travel times are compared to the real ones. Model changes are automatically derived from the travel time residuals. This procedure is repeated based on the changed model. The complete process stops, if distinct stopping criteria are fulfilled, e.g. if the max. number of iterations is reached (see below).

The distancedimension is always METER or FOOT(see global settings menu, chap. 1.2.1), the timedimension is either ms or ns (wavetype is set to electromagnetic wave propagation).

If no model has been defined a start model will be generated automatically when starting the tomography. For that purpose a mean velocity as well as the min./max. coordinates are calculated from the loaded traveltimes. Of course it is also possible to define a start model manually. In this case the model size must be defined in such a way that all wanted shot-receiver-positions are located within the model. In addition, the physical parameters must be defined (p-velocity or ϵ - see below). The simplest model is a homogeneous one, which is a sufficient approximation for many cases. In the case of a refraction tomography it might be useful to include a strong vertical gradient into the start model.

This start model must be transformed into a rectangular grid analogous to the FD-scheme, i.e. it has to be rastered both in x- and z-direction according to the entered space increment. For the seismic wave propagation only the p-velocity (wave type acoustic) is considered. If s-velocities should be used, nevertheless use the acoustic wave type and enter the s-velocities under the v_p option. For the electromagnetic wave propagation the given values of ϵ are automatically transformed into a velocity v according to $v=0.3/\sqrt{\epsilon}$ [m/nsec].

The resulting information-file (inf-file) contains some informations about the tomographic inversion, e.g. the number of iterations, the number of rays and the data residual for the iteration steps. The calculated data residual is the total sum of all residuals of the individual rays. You may divide this value by the number of rays in order to get the mean residual value for each traveltimes. The individual residuals for each ray are calculated as the square of the differences of the real and the calculated traveltimes divided by the datavariance which is given by the parameter "def. data-variance". The datavariance is a measure for the quality of the real traveltimes. The higher this value the lower is the quality. Therefore the individual residuals will be decreased with increased given datavariance because of the higher uncertainty of the real data. The parameter "Desired Statistical Range of Data Residuals" is described under the stopping criteria (statistical criterium).

In addition the total absolute timedifference, the total timedifference and the RMS deviation are written out. The total absolute timedifference defines the sum of the absolute timedifferences independently from the sign (pos. or neg. difference). This value gives you an estimate of the overall adaptation. The total timedifference defines the sum of the timedifferences taking into account the sign of the differences. This value gives you an estimate whether the mean model leads to too small or too big traveltimes, e.g. an interface is too high or too deep. The RMS deviation defines the Root Mean Square deviation.

To stop the iteration, one of the following 5 stopping criteria has to be fulfilled:

Non-physical criteria, i. e. the algorithm is stopped in any way by these so-called knock-out-criteria:

- max. iterations specifies the maximum number of iterations before stopping. (Default value: 10)
- max. time is the maximum CPU-time in sec for the tomography. (Default value: 1000)

Physical criteria:

- threshold is a normalized threshold stopping criterion. If the ratio of the current and of the preceding decrease of the travel time residual is smaller than this value, the iteration process will be stopped. (Default value: 0.001)
- statistical criterium: Setting this option activates a stopping criterium which is related to the variance of the travel times given by the parameter "def. Data-variance". The individual residual for each ray is calculated as the square of the difference of the real and the calculated traveltimes divided by the datavariance. If the difference of the real and the calculated traveltimes of each ray fits the standard deviation, i.e. the square of the difference fits the data variance, the residual of each ray is simply 1 and the sum of all residuals is simply the number of rays. The errors of the real traveltimes are assumed to be Gaussian distributed. For a model that satisfies the data, theory demands a range for the sum of all residuals between number of rays - $\sqrt{2 \cdot \text{number of rays}}$ and number of rays + $\sqrt{2 \cdot \text{number of rays}}$. This is the statistical criterium. These values are listed within the inf-file under "Desired Statistical Range of Data Residuals" (minimum und maximum). Setting this option activates a stopping criterion which is related to the standard deviation of the travel time residuals.
- If the SIRT-algorithm does not converge, the option convergence search controls the maximum number of model changes before stopping the tomography (see below). (Default value: 5).

The SIRT-algorithm calculates the best model changes by theory at every iteration step. In practice it may be advantageous to scale this model changes. In most cases a decrease of the model changes leads to a steadier approach to the final model by the cost of more iteration steps needed.

If the SIRT-algorithm converges the model changes are multiplied with the option modelchange A. A value of modelchange A smaller than 1 decreases the model changes, a value greater than 1 increases the model changes. The value of the option modelchange A has to be positive.

The SIRT-algorithm does not always converge, i. e. it is not able to find a better fitting model with every new iteration step. If no convergence occurs after a certain iteration step, this iteration step is rejected and the preceding step is repeated using model changes multiplied with the option modelchange B. A value of modelchange B smaller than 1 decreases the model changes, a value greater than 1 increases the model changes. In most cases it is appropriate to choose a value smaller than 1 to decrease the model changes. The value of the option modelchange B has to be positive. If the iteration does not converge with this new start model, the procedure is repeated, i.e. the iteration step is rejected and the preceding step is repeated using model changes multiplied with the square of the option modelchange B etc. The iteration process is stopped at the latest if the maximum number of model changes is reached or in advance if a stopping criterion is fulfilled.

PLEASE NOTE: The restriction to small model changes can be useful to get a better final model, but the number of iterations increases with decreasing model changes.

max.def.change(%): allows to enter the default maximum change in percent of the model values from the start model. The default value is 50 %. If you expect higher variations of the model you should set a greater value, e.g. 100 or 200 % (e.g. for refraction data). The parameter should be set in such a way that the resulting parameters are within realistic bounds (e.g. epsilon should not be greater than 80 or smaller than 1).

based on iter. model: if activated the max. change value refers to the actual iterated model. If deactivated the max. change value refers to the parameters given within the start model.

def. data-variance: The default variance of the traveltimes data is entered in the given dimension. The default value is 0.01. Increase this value if the data quality (e.g. noise) is bad. Increasing this parameter results in a smaller residual value for the tomographic result and the stopping statistical criterion will be fulfilled faster. A possible value is the time increment of the original data.

max. beam width: enter the max. size of the beam range in gridpoints.

start curved ray: enter the first iteration number when the curved ray algorithm shall be used instead of the straight rays - precondition: activated option curved rays. Set this value to 1 for the tomographic inversion of refraction data.

average x: enter the x-range in gridpoints over which an averaging shall be done after each iteration step. For the 3D-tomography the entered value average x will also be used in y-direction.

average z: enter the z-range in gridpoints over which an averaging shall be done after each iteration step.

min.velocity: Setting a value > 0 restricts the min. value for the inverted velocities to that value.

max.velocity: Setting a value > 0 restricts the max. value for the inverted velocities to that value.

No check range: allows to set a range (in meter) on the left and on the right where the option Check no ray area is not valid. This might be useful for example for refraction data if far shots are available and here no data points are available between the shot position and the first (last) receiver position. By setting these ranges the start model will not be changed if not covered.

Check no ray area: With this option activated a model area, which is not covered by rays, is separately considered. The model changes in those ranges are calculated from the given changes of the neighbouring cells depending on the setting of the averagex and averagez values. The relation of averagex and averagez defines the preferential direction for the search.

beam: with this option activated the model changes based on the traveltimes residuals are not restricted to the single rays but to a range around the ray. The size of the range is determined from the current traveltimes and the standard deviation of the traveltimes (see also def. data-variance). The option is not available for 3D-tomography.

weighted beam: the option is only enabled if beam is activated. With this option activated a weighting factor within the beam range based on a normal distribution is used for building up the mean value of the model changes within the beam range. The option is not available for 3D-tomography.

curved ray: if activated the curved ray algorithm is used for calculating the traveltimes instead of the straight ray algorithm. The option start curved ray defined the first iteration number for the application of the curved ray algorithm. For a tomographic inversion of refraction data you must activate this option. Note: The 3D-tomography only works with straight rays.

force 1. Iter.: if activated the model after the first iteration is always taken as the start model for the next iteration even if the data variance has been increased. Activate this option if you want to force the tomographic algorithm to change the start model in any case.

fix 1. layer: if activated the velocities for the first (uppermost) layer will not be changed during the tomographic inversion.

restrict to covered area: if activated the velocities outside the area covered by rays will be set to 0. When plotting the result with activated `plotoption autointerpolation` the `plotoption no interp.f.min.color` should be activated in order to disable the interpolation for the borders of the covered area.

restrict to max. angle °: If activated only the pick data with an incidence angle smaller than the given one will be considered.

The option **load data** allows the determination of the input travel time data set. These data have a special format and should be stored in the directory ASCII of the current project directory with the extension TOM. The distance dimension is always METER, the time dimension is either ms or ns (wave type is set to electromagnetic wave propagation). The travel time data may be built up in the module 2D-dataanalysis using the Pick Menu item (see chapter 1.12.2). The geometry is taken from the corresponding trace headers. It is possible to choose several files. If the number of chosen files is greater than 1 the program automatically creates a new ASCII-file named DUMMY.TOM in which all the data of the chosen files are stored.

The tomography input data are ASCII-data. Either a 2D- and a 3D-format is supported. The program automatically controls which format is present.

For the 2D-tomography dataformat one line contains all informations for one travel time:

travel time, code, transmitter_X, transmitter_Z, receiver_X, receiver_Z

The code is given as an integer value.

example (5 travel times):

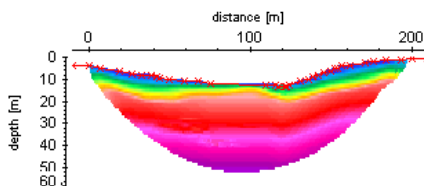
800.00	1	0.00	2.00	100.00	2.00
801.60	1	0.00	2.00	100.00	6.00
803.20	1	0.00	2.00	100.00	10.00
806.20	1	0.00	2.00	100.00	14.00
811.00	1	0.00	2.00	100.00	18.00

For the 3D-tomography dataformat again one line contains all informations for one travel time:

traveltime,code,transm._X,transm._Y,transm._Z, receiv._X, receiv._Y, receiv._Z

example (5 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		
803.20	1	0.00	2.00	1.00
100.00	10.00	5.00		
806.20	1	0.00	2.00	1.00
100.00	14.00	5.00		
811.00	1	0.00	2.00	1.00
100.00	18.00	5.00		



The radiobox `sec.coord.` (see chap. 4) controls which second coordinate (y or z) is used for the 3D-tomography dataformat if a 2D-tomography will be performed.

The first coordinate is always x. The third coordinate is neglected. The program gives a warning message if only 0 values are present within the tomographic data for the chosen second coordinate and changes the `sec.coord.` if wanted.

The tomography always works with depth values. Therefore if the original z-values represent altitudes you must activate the option **altitude** and enter a reference value. The option must be active before the tom-data will be loaded. Then the original altitude values within the tom-file will be transferred to reference level - altitude values..

The geometry (transmitter_X ...) are in meter, the travel times are in msec (seismic), nsec (electromagnetic) respectively. The code has no significance at present and should be set to 1.

If the option **topography** has been activated (see chap. 4.4.6) all source and receiver positions above the topography are automatically placed directly below the topography. Therefore it is not necessary to redefine the z-positions of the source and the receivers if a topography will be used in addition.

With the option **create topography from data** activated the first layer of the start model will be automatically modified based on the topographic z-values contained within the traveltime data loaded for the tomographic inversion. The option must be activated before loading the tomographic data (option load data). The option **topography** (see above) will be automatically activated. If altitude is activated the z-values must represent altitude values and will be transformed into depth values using the entered reference level. The resulting topographic interface may be still manually changed (e.g. using the option hor. extrap.) before performing the tomography. It is recommended to activate this option for a seismic refraction inversion if the topography shall be read from the tomographic data because in this case it is ensured that no rays will penetrate the area above the topography. Normally it is not useful to activate the option for a transmission tomography.

After having loaded the data the program automatically sorts the data after ascending source and receiver positions and stores the data under dummy.tom under the actual project directory using 3D-data tomography dataformat. Therefore it is not recommended to use the name dummy.tom for the tomography data.

If the option **view data** is activated the traveltime data are displayed. The data are displayed along the given projection axis (default x - see also traveltimes radio box). Therefore it is only recommended to view the data if they have been acquired along one of the coordinate axis (e.g. seismic refraction data).

With **final model** you may enter the filename for the final model. The final model is automatically converted in REFLEX format and stored in the directory ROHDATA of the current project directory with the extension DAT. The final model contains velocities in m/sec for the seismic wave propagation and the values of ϵ in the case of the electromagnetic wave propagation.

In addition, an ASCII file with the extension CON is generated, which contains the sums of the ray lengths in each grid cell giving an idea of the confidence of the calculated grid velocities.

At last, an ASCII info file with the extension INF is generated.

The option **3D-tomography** within the 3D-tomography box activates the 3D-tomography. The modelsize in y-direction must be entered within **ymin** and **ymax**. If the 3D-tomography is activated the data must have the 3D-dataformat specified above under the option load data. The data-format of the final model is the

REFLEX 3D-dataformat. The several X-Z cuts for each y-point are stored behind each other within one file. It is possible to use the 3D-datainterpretation possibilities to look for the different 2D-cuts.

DoRaster: The option Raster allows the rastering of the model parameters based on the current space increment. In any case the rastering should be done before starting the Tomography. The 2D-arrays of the individual model parameters are stored on different files with the following filenames: model file name + special extension. The different extensions are:

electromagnetic wave propagation: ε - FDP;

seismic wave propagation: Vp - FDP;

Storing the arrays is done in an ASCII-format using the following block notation (all z-values for one x-point):

FDP-files: 10F8:2

At the end of each block the line must be ended.

example - 15 values in z-direction, 2 blocks = 2 successive x-values:

```
10.00 10.00 10.00 10.00 10.00 10.00 10.00 10.00 12.00 12.00
12.00 12.00 12.00 12.00 12.00
10.00 10.00 10.00 10.00 10.00 10.00 10.00 10.00 12.00 12.00
12.00 12.00 12.00 12.00 12.00
```

start tomography starts the tomography. After activating this option the chosen model is rastered based on the given parameters and then the tomography is started. During the computation the ProcessingStatus is shown in a window. The resulting 2D-velocity-file of that iteration step preceding the current is displayed in another window, too.

After the algorithm is stopped by one of the stopping criteria the final 2D-velocity-file is displayed in this window and the current stopping criterion together with the number of iterations is shown. Please click on the close button to finally stop the memo window.

With the option show result activated the final model is displayed within the start model image.

With the option confidencefile activated a Reflex formatted confidence file is generated which may be displayed for example within the 2D-dataanalysis - the filename is queried after having finished the tomography. The confidence is a measure for the number of rays within one cell. With the beam option activated the beamrange around the ray is taken into account in addition.

An information file with the extension inf will be automatically created. It has been described at the top of this chapter.

The option **raster** controls if the current model is rastered with the given deltaX or if an already existing raster file will be used. With the option raster deactivated you are asked for the raster file (extension fdp). In this case the chosen raster file must have the same size as the raster file which would be generated from the current model with the given deltaX. Deactivating the option raster might be useful for example in order to remake a tomographic inversion based on a previous tomographic result.

check rays: After having chosen this option an ASCII-file is generated for all the rays containing the following values in each line:

time SourceX SourceZ RecX RecZ distance mean_velocity

The file gets the name CHECKRAY.REP and is stored under the path ASCII.

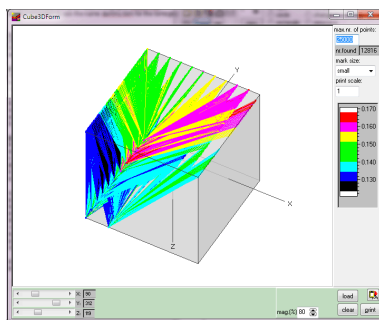
show rays: two cases will be considered dependent on the setting of the option 3D-tomography.

2D-tomography: The option show rays allows the display of the rays based on the currently loaded ASCII-TOMO traveltimes data into the start model. Every n.ray will be displayed. The rays are displayed as straight lines from the source to the receiver. Therefore the option is not suitable if curved rays shall be interpreted, e.g. for the refraction tomography.

If the option colored within the ShowRays Box is activated the rays are plotted with different colors. The color-code corresponds to the mean velocity of the ray and is based on the current loaded color palette (see plotoptions chap. 1.7.2). The following values must be input: min. and max. velocity for the color code (velocities ranging outside this interval are plotted in the first or last corresponding color). Default: the values are automatically determined from the loaded traveltimes data. One goal of the color code is to realize possible errors more easily. If the option ShowOnlyShotRec is activated only the positions of the shots and the receivers are viewed when clicking on show rays.

3D-tomography: With the option show rays activated the rays are plotted within a separate 3D-cube window.

With the option colored active the rays are plotted with different colors (rainbow colors with red corresponding to the max. velocity and dark blue to the min. velocity). The color scale within the 3D-cube menu is only available within the Reflexw "c" version using the Windows themes.



min/max. xz(y): allows to extract the min./max. border values from the loaded tomographic "tom" traveltimes data.

BatchStartTomo: allows to make several tomographic inversions within one step using different tomographic parameters. After having started the option an ASCII batch file is queried which must have the following structure - example with 3 different parameter sets:

```
iterations: 10 12 14
threshold: 0.001 0.003 0.002
modelchangeA: 1 0.9 0.8
modelchangeB: 0.1 0.11 0.12
convergenceSearch: 5 6 7
maxdefchange: 200 150 100
basedonitermodel: 0 1 0
defdatavariance: 0.01 0.02 0.03
maxbeamwidth: 10 12 8
averagex: 0 2 4
averagez: 0 1 2
statisticalcriterium: 1 0 1
checknorayarea: 1 1 0
beam: 1 0 0
weightedbeam: 0 1 1
force1iter: 1 0 1
```

The final models will be saved under the given final model name together with a running number (for this example final_model1 final_model3).

4.7 raytracing/wavefront inversion

If the RayGroupBox is activated (e.g., use the speed button RAY in the first lower panel), the parameters necessary for the raytracing and the traveltime inversion can be entered. The rastering of the model based on the current parameters and the raytracing/traveltime inversion must be started from this box, too.

Within this group box you have the possibility of calculating **synthetic traveltimes** using two different raytracing methods (see raytracing 2D, chap. 4.7.1) and to invert picked forward and reciprocal traveltimes using a **wavefront inversion technique** (see wavefront-inversion 2D, chap. 4.7.2). Both the method to calculate traveltimes and the wavefront inversion technique are suitable for the interpretation of **first arrivals**, e.g. refraction traveltimes and they are restricted to 2D. The simulation of the synthetic traveltimes can also be done for **primary reflections** (only 2D, see chap. 4.7.1.2) and for a 3D-medium (only first arrivals-see chap. 4.7.1.4). The wanted method is chosen within the **type** option.

DoRaster: The option Raster allows the rastering of the model parameters based on the current space increment. For more detailed information see FD-model simulation, chap. 4.5).

reflex rasterfile: if activated a Reflexw formatted rasterfile using the current raster parameters will be generated in addition when using the option DoRaster. The filename is taken from the modelfilename and the rasterfile is stored under rohdata under the current project directory. The file may be viewed under the 2D-dataanalysis. If the option 3D is activated a 3D-Reflexw rasterfile is generated which may be viewed within the 3D-datainterpretation.

start starts the rastering and then the raytracing or the wavefront-inversion. After having finished the raytracing the calculated traveltimes are shown in the lower window. The upper window shows the model together with the optional rays. After having finished the wavefront-inversion the current model is updated.

Adapt reflector: performs an automatic correction of the reflector depth by comparing the calculated and the synthetic reflection traveltimes (see chap. 4.7.1.3).

4.7.1. raytracing 2D

The calculation of the **synthetic traveltimes** is restricted to the first arrivals and the primary reflections for an arbitrary 2-dimensional medium. No secondary arrivals can be calculated. This can be done using the FD-simulation (chap. 4.5). The wanted method is chosen within the **type** option. Two different methods are available.

With the **network raytracing** the raypath with the shortest traveltime from one point to the other is determined. This method only allows the simulation of first arrivals for both 2D and 3D models (option **3D-ray** - see chap. 4.7.1.4).

As a prerequisite for the calculations the model has to be gridded with the appropriate spatial increment **DeltaX**. The next step is to develop a so-called Forward-Star. The Forward-Star of a node i is the set of nodes which are connected directly with the node i .

The calculation of the shortest raypath is done with the Dijkstra algorithm. At the beginning of the computation the traveltime for the source-node, which is the location of the point-source, is given. In order to determine the traveltime to the other nodes, the Dijkstra algorithm is applied, which won't be described in more detail, in here. The interested reader may refer to the corresponding specialized geophysical literature. For optimizing the computation time various changes (according to Klimeš & M.Kvasni ka) are implemented. The accuracy of the calculations depends on the order of the spatial increment, with which the model has been gridded and on the size of the Forward-Stars. The amount of grid-points is restricted to 150000 points.

The option **calculate** controls if one single line or several lines are calculated. For detailed information please refer to raytracing shot/receivers, chap. 4.7.1.1.

With **FD-vidale** a method based on a finite difference approximation of the eikonal equation for calculating synthetic traveltimes is used. It takes into the account the existence of different propagation waves like transmitted, diffracted or head waves. Therefore no practical limitation concerning the complexity of the medium is given. Again the model has to be gridded with an appropriate spatial increment **DeltaX**. Two different FD-algorithms are available. With the option **Podvin** activated a more accurate but also more CPU-time consuming method is used. Starting from the source point traveltimes are calculated in a quadratic region. If headwaves are detected the propagation direction must be redirected in order to find the minimum traveltimes.

The option **iterations** controls the number of redirections of the wavefront propagation in order to take account possible additional head waves. Increasing this number also significantly increases the CPU-time.

The option **calculate rays** controls if the rays are calculated in addition to the traveltimes or not. The option **exact ray** controls how the rays are calculated. With exact ray activated always the steepest descent of the local best fitting plane to the neighbouring traveltimes is calculated. With exact ray deactivated the rays are always calculated from the greatest decrease in traveltime of the neighbouring gridpoints. It depends on the model which method gives the better results. The option **ray discretization** describes how the rays are discretized. Low means every 4. sample point, med. every 2. and high each sample point is used.

Activating the option **smooth rays** enables a smoothing of the rays.

The option **reflections** allows the simulation of primary reflections from the given model boundaries - see chap. 4.7.1.2. An automatic correction of the reflector

depth by comparing the calculated and the synthetic primary reflection traveltimes from a special boundary is also available (option adapt reflector see chap. 4.7.1.3).

The option **calculate** controls if one single line or several lines are calculated. For detailed information please refer to raytracing shot/receivers, chap. 4.7.1.1.

The given of **output-scale** controls, together with DeltaX, the sampling in x- and z-direction of the output.

The option **calculate rays** controls if the rays are calculated in addition to the traveltimes or not.

The option **raster** controls if the current model is rastered with the given deltaX or if an already existing raster file will be used. With the option raster deactivated you are asked for the raster file. Two different filetypes are supported: fdp-rasterfiles (stored with the extension fdp when using the option doraster) and reflex-files (*.??t). In case of no rastering the chosen raster file must have the same size as the raster file which would be generated from the current model with the given deltaX. Deactivating the option raster might be useful for example in order to perform a raytracing for a tomographic inversion model (see also chap. 0.10.4.3).

With the option **syn.distances** activated the stored distances from the loaded data traveltimes are taken over for the calculated traveltimes. This might be useful if you want to load the calculated traveltimes into the original seismic data within the 2D-dataanalysis module.

4.7.1.1 raytracing shot/receivers

The calculate option allows to set the wanted type of calculation:

single line: The first arrivals and the primary reflections (optional) for a single line are calculated and stored in REFLEXW pick format using the outputfile name under the path LINEDATA under the current projectdirectory. You have to specify the coordinates of the source (sourcex, sourcez) and of the receiver line (recxstart, recxend, reczstart, reczend). The given of output-scale controls, together with DeltaX, the sampling in x- and z-direction of the output.

several single lines: This option allows to compute a sequence of different single lines based on the current model. Only first arrivals are calculated. The given of output-scale controls, together with DeltaX, the sampling in x- and z-direction of the output.

Up to 999 different single shots may be simulated in one step.

An ASCII file FDSHOTS.* (FDSHOTS.1, e.g.) under the path MODEL must exist which contains in each line for a single modelling the shot, receiver positions, respectively, and the name of the file, in which the traveltimes shall be stored:

sourcexpos sourcezpos recxstart recxend reczstart reczend traveltimename

The number of lines in the file controls the number of different calculations.

example:

```
0.0 0.0 0.0 45.0 0.0 0.0 test10
10.0 0.0 0.0 45.0 0.0 0.0 test20
20.0 0.0 0.0 45.0 0.0 0.0 test30
30.0 0.0 0.0 45.0 0.0 0.0 test40
40.0 0.0 10.0 45.0 0.0 0.0 test50
```

Five simulations with different source and receiver positions will be generated.

The resulting five single lines will be stored with different names in REFLEXW pick format in the directory LINEDATA of the current project directory (test10.pck, test20.pck, test30.pck, test40.pck, test5.pck, respectively).

After starting the computation (**Start** button) the ASCII file FDSHOTS.* has to be chosen whereupon the included shot geometry is displayed.

Exploding reflector: allows to simulate the zero offset traveltimes from the individual reflectors within one step (see chap. 4.7.1.2).

Data traveltimes: This option allows to calculate synthetic traveltimes at those distance positions where data traveltimes are displayed in the lower window panel (option load datatraveltimes, chap. 4.1). The x-positions of the shots to be calculated are automatically taken from the shot- and receiver geometries of the currently loaded data traveltimes (see also option load datatraveltimes, chap. 4.1). The z-positions depend on the actual settings (see below). For the setting "value" the following holds true:

- the z-positions of the shots and of the receivers are taken from the traveltimes depending on the setting of the sec.coord. radiogroup (either the y-coord. or the z-coord. of the traveltimes will be taken into account).

Please consider: If the z-positions are altitudes the option altitude must have been activated before loading the data traveltimes.

The given of output-scale controls, together with DeltaX, the sampling in x- and z-direction of the output .

Up to 99999 different single shots may be simulated in one step. If the checkbox near the outputfile is activated only one single traveltimes and ray outputfile will be created containing all shots (only activatable if no reflections are calculated). If deactivated for each shot such a traveltimes and ray outputfile will be generated. Then the filenames will be automatically determined from the entered outputfilename (the first 4 characters) and the shot positions (either the x-positions together with the label 'x' or the y(z)-positions together with the label 'z'). For the shot positions only the integer part will be used together with a multiplication factor if DeltaX is smaller than 1 (factor 10, 100 or 1000). If a filename conflict occurs (e.g. if the shots are located at the same x- or y-position but in different y(z)- or x-positions like at opposite sides of the medium) the filenames will be automatically determined from the shot numbers.

The source and receiver **z-positions** may also be located on any interface. On the right hand side of the "source z" option you may define if this value shall be taken or if the source shall be located on the 1. layer (topogr.) or on the 2.-5. layer. Equally the option next to the "rec z-start" and "z-end" options allows you to define the receiver line according to the entered values or to the chosen interface (1.layer(topogr.) or on the 2.-5. layer). Depending on the setting of the option topography (see also model generation - topography, chap. 4.4.6) "value" (not activated topography) or "topogr." is highlighted by default both for the source and the receiver line.

The option **create several lines file** allows to generate the ASCII-file which is necessary for the ASCII-file of the option several lines file. The parameters places and decimal places within the global settings menu are used for the format of the ASCII-file.

Activating **fixed line** allows you to define the geometry for a fixed receiver line for different shot points. The right panel shows an example of shots located from 0 (source z start) to 100 m (source z end) with a spacing (source incr.) of 5 m within a vertical borehole at 0 m (source x start and source x end). The receivers are located from 0 (rec. z start) to 100 m (rec. z end) within a second vertical borehole at 60 m (rec. x start and rec.x end).

Activating **moving line** allows you to define the geometry for a receiver line moving with the shots. The receiver start and endcoordinates will be updated for each shot using the entered source incr. The input parameters are the same as for the fixed line.

create SeveralLines file

☒ fixed line
☐ moving line

source x start: 0
source x end: 0
source z start: 0
source z end: 100
source incr.: 5
rec x start: 60
rec x end: 60
rec z start: 0
rec z end: 100

filename: ver
file extension: 0

generate close

4.7.1.2 raytracing reflections

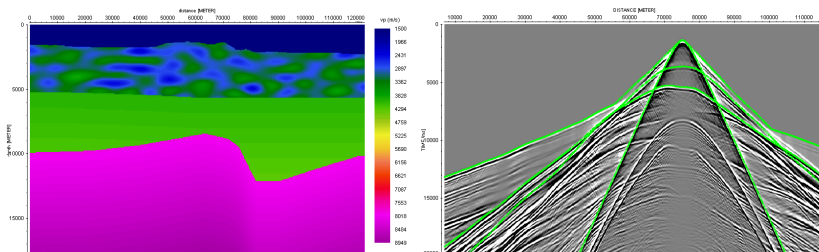
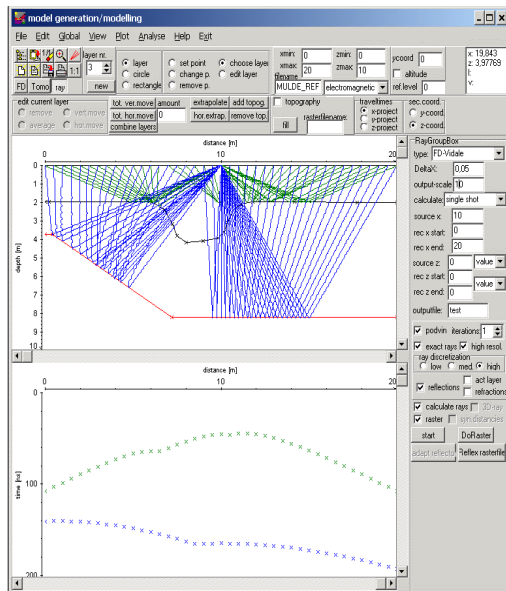
With the option **reflections** activated the primary reflections from the individual layers or only from the actual red undelined layer (option **act.layer** active) are calculated in addition using the FD-Vidale raytracing. The traveltimes are stored under a pck-file with the filename corresponding to the entered outputfile plus a characterization of the layer number. Example: outputfile TEST - the reflection from layernr. 2 are stored under: TESTR2.pck.

The direct refracted traveltimes are calculated in addition if the option **refractions** is active.

The **rays for the reflections** are calculated if the option **calculate rays** is active. An addition option **high resol.** controls if the calculation of the rays is done using a high resolution. This leads to a larger computation time.

Reflections can be calculated for the **calculate types** single shot, data traveltimes and exploding reflector.

The layershow colors are used for discriminating the refractions and the reflections from the different layers if the calculate type **single shot** has been used.

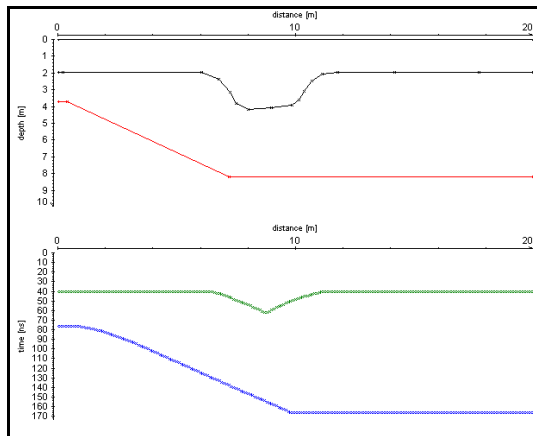


Synthetic seismogram (Finite Difference approximation of the waveequation) with underlined calculated traveltimes (refractions and reflections) for the model shown on the left. A good match between FD-simulation and calculated traveltimes has been achieved both for the refractions and the reflections.

With **data traveltimes** activated and only one shot loaded again the layershow colors are used in the same manner like for single shot.

With data traveltimes activated and different shots loaded the layershow colors are used in order to discriminate the different shots. Refractions and reflections from the same shot use the same color. The rays are only plotted for the lowermost reflector.

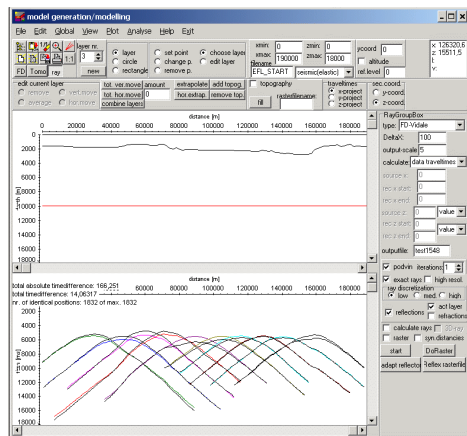
Activating **exploding reflector** allows to simulate the zero offset traveltimes from the individual reflectors within one step. Again the layershow colors are used for discriminating the reflections from the different layers (see figure on the right). The option reflections is always active and the option refractions always deactive. The option act. layer allows to simulate the zero offset traveltimes only from the actual layer boundary.



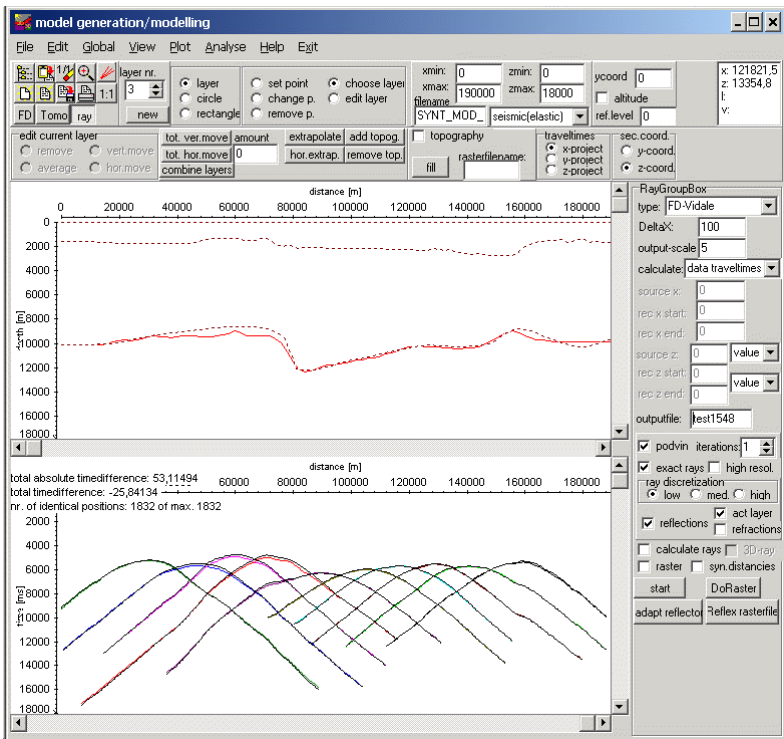
4.7.1.3 automatic adaptation of primary reflections

The option **adapt reflector** allows an automatic correction of the reflector depth by comparing the calculated and the synthetic reflection traveltimes. Before applying this option you must:

- load or create a model including the reflector boundary which shall be adapted. The overburden will not be changed - therefore the overburden must represent the real conditions as well as possible, e.g. after having performed a wavefront inversion of the first arrivals or after having done an interactive raytracing. It is also possible to use a tomographic result. For the latter purpose the option raster must be deactivated and the same rasterincrement DeltaX and the same border coordinates must be used as for the tomography (procedure is the same like for the "normal" raytracing).
- load the data traveltimes which shall be adapted using the option file/load datatraveltimes
- choose the wanted layer boundary (underlined in red)
- choose the calculate type data traveltimes
- activate the options reflections and act. layer and deactivate the option refractions
- The option calculated rays should be deactivated in order to save computer time
- enter the rasterincrement DeltaX and start the ray tracing in order to get a first overview (optional) check. The option calculate traveltimes under Analyse can be used in order to calculate the traveltimes differences between real and synthetic data.
- activate the option adapt reflector. Now the number of iterations is queried (default: 5). After having entered the iteration number the adaptation starts. The model (the actual layer) is always updated after each iteration step and the adaptation parameters are shown. After having reached the number of iterations or if no improvement has been achieved any more the adaptation stops. A report file adaptation.rep (stored under model under the project directory) will be automatically created indicating the iteration number and the traveltimes differences and the coverage of the individual reflector points. The coverage will also be displayed as a grayscale bar at the bottom of the model. This bar will be displayed as long as the button adapt reflector is highlighted. Click again on button adapt reflector in order to deactivate it.



The following picture shows the result after 10 iterations using the start model shown within the upper picture. The true model is underlined using dashed lines.



4.7.1.4 3D raytracing

The option **3D-ray** allows a 3-dimensional raytracing based on the network raytracing.

The parameters **ymin** and **ymax** defines the model range in y-direction. The rasterincrement DeltaX also holds true for the y-direction.

The parametera **source y** and **receiv. y** define the source and receiver positions in y-direction. Restriction: the receivers may only be located along one constant y-position.

The 3D-model is built up from different 2D-cuts lying within the xz-plane. The parameter ycoord defines the location of each cut in the y-direction. The option **load** allows you to define the wanted 2D-model cuts.

4.7.2 wavefront-inversion 2D

Activating **wavefront-inversion** allows to migrate the picked forward and reverse traveltimes into depth using a FD-approximation of the eikonal equation analogous to the forward raytracing FD-vidale method. Again the model has to be gridded with an appropriate spatial increment **DeltaX**. This value should be chosen in such a way that no significant inhomogeneity of the overburden will be lost. The computertime for the inversion increases by a factor of 4 if you decrease the increment by a factor of 2. The complete forward and reverse wavefronts are continued downward based on the given overburden model. The new refractor is constructed at those points where the sum of the downward traveltimes is equal to the reciprocal traveltimes. The refractor velocity is determined from the mean of the slopes of the forward and reverse wavefronts at the new calculated refractor points.

Precondition for the application of the wavefront-inversion is the existence of both a complete **forward** and **reverse** traveltimes curve (see also traveltimes analysis and traveltimes combine, chap. 5.3). Only these traveltimes must be used which have been stored within the combine traveltimes panel.

The overburden must be existent. It may contain any 2-dimensional structure. After having entered the parameters (deltaX and outputscale) the wanted traveltimes datafile must be chosen. If the wavefrontinversion module is called from the traveltimes analysis menu the current combined traveltimes curves are used and no datafile must be chosen. The traveltimes curves are automatically reinterpolated based on the given DeltaX. The CPU-time drastically increases with decreasing DeltaX. Therefore the best compromise between needed resolution and CPU-time must be found. A window informs you about the status of the calculation.

The parameter **nr. of diff. velocities** controls the number of different refractor velocities to be determined. After having finished the wavefront inversion the new window "**velocity determination using linear regression**" opens showing the resulting forward and reverse traveltimes for the determination of the refractor velocities. These traveltimes branches are automatically subdivided into a number of different linear regression curves dependent from the chosen number of different velocities. The resulting velocities are shown in the middle of each regression curve. You may change the start and end position of each regression curve by simply clicking on it and drawing it with pressed left mouse button. The forward and reverse branches are connected to each other. Therefore for example any change on a forward regression curve also acts on the equivalent reverse regression curve.

Activating the option **close** terminates the velocity determination window and the new new layer boundary is plotted. The refractor velocities are determined from the mean of the two velocities for the forward and reverse branches.

The inverted interface should be extrapolated either to the vertical model borders (use option **hor.extrap.**) or to the nearest layer or model border (use option **extrapolate**).

5. Refraction traveltimes analysis 2D

Within this menu it is possible to analyse and interpret picked first arrivals (refraction seismics). The distancedimension is always METER or FOOT (see global settings menu, chap. 1.2.1), the timedimension is either ms or μ s. Precondition is that all chosen data are located along **one line** within one acquisition plane. You may put together the traveltimes from several shots (traveltimes put together, chap. 5.1) and assign the traveltimes to special layers (option **assign**, see also traveltimes assign, chap. 5.2). Those traveltimes stemming from several shots and belonging to one layer may be combined together to one forward and reverse traveltimes curve (option **combine**, see also traveltimes combine, chap. 5.3). These combined traveltimes are the basis for a subsequent wavefront-inversion 2D (chap. 4.7.2) which allows to invert both the structure of the layer and the smoothed refractor velocity (option **wavefront-inversion**).

The following options are always valid:



enter the project directory



copy current traveltimes picture to the clipboard



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



enable manual zoom - With the option ZOOM an arbitrary area of the data set can be selected and plotted in full screen size



Print traveltimes

The traveltimes curves are always marked with shot numbers, beginning with 1, in ascending order corresponding to ascending shot coordinates.

show traveltimespoints under view: allows to display or to hide the traveltimes points.

With the option show traveltimelines under view activated the traveltimes belonging to one shot and to one layer (activated option assign or combine) are joined by a straight line.

With the option shot number under view activated the shot number corresponding to the traveltimes pick nearest to the current mouse position is shown within the coordinate window in addition.

With the option showxyprojection under view the xy-positions of the current picks are displayed. The option is useful for picks with GPS coordinates for a direct control of the locations of the picks within the xy-plane and can be used in order to check the pathway and the direction of the acquisition line for e.g. a later projection on the x-axis (see option project on x-coord. under edit).

The option **plot options** under **plot** allows you to enter some plot options like show grid, etc.

The speedbutton **V** or **velocity adaptation** under **analyse**: allows to determine a velocity by creating a straight line with two mouse clicks. With this button activated the program waits for the input of two datapoints for generating a straight line of distinct velocity. Pressing the left mouse button determines the first datapoint and by moving the mouse with pressed button the line with the current velocity is shown. Leaving the mouse button holds the current line.

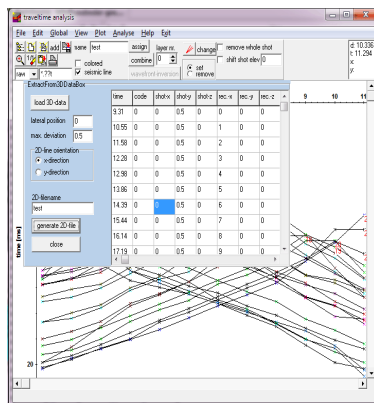
The option **load second traveltimes** under **file** allows to load secondary traveltimes, which are plotted as comparative traveltimes in addition to the current ones. You may choose different pick files from the file list. These traveltimes or secondary picks are displayed in red rectangles. The option is convenient to control traveltimes of neighbouring shot points or to compare real traveltimes with calculated ones (see option calculate traveltime differences under **analyse**). The option **second traveltimes** under **view** controls if the secondary traveltimes are visible or not.

The option **export to ASCII** allows to save the currently loaded traveltimes into an ASCII-format. The format corresponds to the ASCII-3D-tomography format within the Pick save Menuitem (chap. 1.12.2.1). Each line contains the following values: traveltime(8:2), code(8:0), x-traceheadercoordinate of the transmitter, y-traceheadercoordinate of the transmitter(8:2), z-traceheadercoordinate of the transmitter(8:2), x-traceheadercoordinate of the receiver(8:2), y-traceheadercoordinate of the receiver(8:2), z-traceheadercoordinate of the receiver(8:2). The file has the extension TOM and will be stored under the path ASCII under the current projectpath. The format corresponds to that format needed by the 3D-tomographic traveltime-inversion.

The option **extract from 3D-data** under **file** allows to extract 2D-lines from a 3D-dataset. The 3D-dataset must have the tomo-3D-coordinates dataformat.

The wanted 2D-lines may be orientated in **x- or y-direction**. The option **lateral position** defines the coordinate perpendicular to the entered orientation. All data lying on this line will be taken when a 2D-line is generated.

The option **max.deviation** allows to enter a deviation value for the lateral shot position. This means the shots may be located outside the 2D-line orientation. The sign of the max. deviation is taken into account. In the example the shots having a shot-y coordinate between 0 and 0.5 m will be used for the generation of the 2D-line.



The option **calculate traveltime differences** under **analyse** allows to calculate traveltime differences between the current and the secondary traveltimes. As the positions of the current and the secondary traveltimes may vary because of the

fixed raster increment of the raytracing you may enter a position bin which is used for determining the quasi-identical positions. The total absolute timedifference, the total timedifference, the RMS deviation and the number of identical positions are displayed. The total absolute timedifference defines the sum of the absolute timedifferences independently from the sign (pos. or neg. difference). This value gives you an estimate of the overall adaptation. The total timedifference defines the sum of the timedifferences taking into account the sign of the differences. The RMS deviation defines the Root Mean Square deviation. The number of identical positions is shown together with the total number of current picks in order to have an estimate over the significance of the analysis.

The option **calculate reverse traveltimes** under analyse allows to control the reverse traveltimes. The reverse traveltime and the relative traveltime error should be checked in any case. Activating the option a new window named "show reciprocal times" appears which allows the following inputs:

output: you may choose a file output or an output to the screen.

With the option **file** chosen an ASCII-file named reciprocal.times under the current project directory is created. One file block (two lines) contains a pair of reverse traveltimes (shot_x, shot_y, receiver_x, receiver_y, traveltime).

With the option **screen** chosen the reverse traveltimes are shown on the screen. Each pair of reverse traveltimes is shown in one color (the layershow colors are used.). In addition to the traveltime the corresponding shot number is shown.

traveltimes choice: you may choose if all traveltimes (option all), only traveltimes for a definable shotnumber (option one shot) or only picks for a definable layernumber (option one layer) are taken into account.

With the option **one shot** chosen you have to enter the wanted shot no. The minimal shot number is 1. Shot numbers are in ascending order according to ascending shot coordinates. The relative traveltime error is calculated from the traveltimes of the selected shot. The relative traveltime error and the reverse traveltime of this shot are displayed.

With the option **one layer** chosen you have to enter the wanted layer no. An assignment of traveltimes to layers prior to this option is required. The relative traveltime error is calculated from all traveltimes assigned to the desired layer. The relative traveltime error and the reverse traveltimes of this layer are displayed.


As a summary the mean reciprocal time error is displayed.

The option **Time Term interpretation** under Analyse allows the reconstruction of a 3D-refractor from xy-traveltimedata (see chap. 5.4).

You have the possibility to show the **corresponding seismic shot section** by simply double click on any traveltime pick if the option **seismic line** is activated. The program automatically searches the default datapath (raw or proc, if proc is chosen you also have the possibility to set any filefilter in addition) and shows the shot section with the same shot coordinates as the chosen traveltime pick within a separate window. If no section could be found you may choose any shot section from the filelist. The picks are also automatically plotted if the picks have been stored under the same filename like the seismic shotfile.

5.1 traveltimes put together

The first step for the refraction traveltime interpretation after having picked the first arrivals within the menu 2D-DataAnalysis (see also Pick Menuitem, chap. 1.12.2) is to put together the traveltimes from the several shots. Precondition is that the shots and the receivers are located along **one line** within one acquisition plane. The chosen traveltimes are automatically sorted after shot and receiver positions in ascending order. Forward and reverse traveltimes are automatically separated. The traveltimes are plotted over the distance. The distance is always determined from the xy-coordinates of the receiver-positions. The start distance is determined from the minimum x- or y-coordinate of the line. The min. x-value is chosen if the pronounced line direction is the x-direction, the min. y-value is chosen if the pronounced line direction is the y-direction. With the option **colored** activated the layershow-colors are used for discriminating the individual shots.

The option **load traveltimes** under file or the speedbutton  allows to load the traveltimes from different pick files. Already existing traveltimes will be lost. Use this option if you want to load traveltimes which have already been put together in a previous step. If only one file is loaded the filename for saving the traveltimes is automatically set to this datafilename. It is possible to load REFLEX formatted PCK files as well as

2D-columns ASCII with each line containing

distance traveltime

ASCII-3D-tomography with each line containing (see also chap. 1.12.2.1)

traveltime code x-transmitter y-transm. z-transm. x-receiver y-receiver z-receiver

ASCII-2D-tomography with each line containing (see also chap. 1.12.2.1)

traveltime code x-transmitter y-transmitter x-receiver y-receiver


In addition you may load ASCII-matrix data (option Shot-Rec.Matrix, m shots and n receivers) which has the following format:

1. line contains the x-positions of all m shots
2. line contains the z-positions of all m shots
- 3.-n.lines:
 - 1.column: x-receiver position
 2. column: z-receiver position
 - 3.-m. Column: picked traveltimes


example:

shot position	-100.00	-50.00	3.75	41.25	78.75	132.50	182.50
GP pos. Elev.	0.00	0.00	0.00	0.00	0.00	0.00	0.00
0.0	0.00	110.00	59.00	10.00	57.00	78.00	140.00
7.5	0.00	115.00	62.00	10.50	50.00	73.00	137.50
15.0	0.00	120.00	65.00	20.00	46.00	69.00	132.50
22.5	0.00	124.00	71.00	28.00	42.50	62.50	130.00
30.0	0.00	128.00	78.00	39.50	25.00	60.00	130.00
37.5	0.00	130.00	94.00	47.00	11.00	50.00	122.50
45.0	0.00	135.00	105.00	56.00	10.50	45.00	120.00
52.5	0.00	138.00	116.00	60.00	22.00	37.50	115.00
60.0	0.00	141.00	119.00	62.50	30.00	31.50	109.00
67.5	0.00	143.00	121.00	68.00	38.00	24.00	102.50
75.0	0.00	147.00	126.00	72.00	45.00	15.00	95.00
82.5	0.00	150.00	130.00	76.00	50.00	9.50	85.00

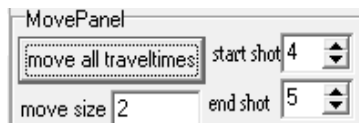
The option **add traveltimes** under file or the speedbutton **add** allows to add traveltimes to the existing ones.

The option **reset traveltimes** under file or the speedbutton  removes all existing traveltimes.

The option **reload traveltimes** under file or the speedbutton **rel**, allows to reload the same previously loaded traveltime pick files. Any changes within these pickfiles will be taken into account. The shot zerotraveltime will also be included if this has been activated before. The option might be useful for a fast control of any changes of the individual pick files if e.g. the seismic profile has been loaded within a second window with activated pick option.

The option **save traveltimes** under file or the speedbutton  allows to save the existing traveltimes under the datafilename specified within the option **filename** (saved with the extension PCK under the path LINEDATA under the current project directory).

The option **change** under analyse or the speedbutton **change** allows to change individual traveltimes. The options move and remove determine the action.



With the option **remove** activated individual traveltimes may be deleted using the left mouse button. With the option **remove whole shot** activated the whole shot will be removed.

With the option **move** activated either a block of traveltimes may be moved in time or all traveltimes for a given set of shots.

Pressing the left mouse button determines the first datapoint. Pressing the left mouse button again determines the second datapoint and by moving the mouse with pressed button all traveltimes lying inbetween these traveltimes are moved. To be considered: the traveltimes are sorted after shotnumbers. Therefore if you are choosing two traveltimes stemming from different shots all traveltimes stemming from the shots lying inbetween these shotnumber will be moved.

Activating the option **move all traveltimes** allows the moving of all traveltimes of one or more shots. Enter the start shot and end shot and the move size. When clicking on **move all traveltimes** the traveltimes of all shots lying between start and end shot will be shifted by the entered move size (given in time dimension, normally ms).

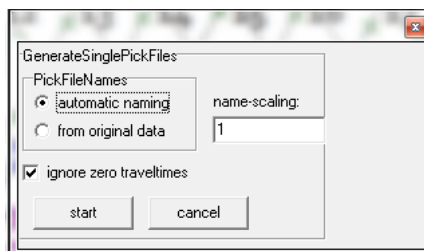
There are some other possibilities of editing the traveltimes under **edit**:

Insert shot Zerotraveltime: If a shot point does not coincide with a geophone position there is no measured traveltime at that shot point. This option inserts a traveltime with timevalue 0 at every shot point. To be considered if the traveltime data contain any elevations: the shot elevations might be different from the receiver or surface elevations (e.g. shot within a borehole). In this case it might be useful to activate the option **shift shots** before inserting the zero traveltimes. The entered mean shift value will be added in order to compensate the different elevation levels.

CMP sort: This option allows to resort the traveltimes after Common Mid Points. The user is asked for a value for the CMP averaging (CMP-bin). This value should be of the order of the average shot point increment. CMP sorted traveltimes are well suited for 1D modeling.

generate single shot(CMP)-traveltimes:

By choice of this option traveltimes already put together from different shots or CMP's may be broken up in individual files. The filenames may be automatic or generated from the original seismic lines. If **automatic naming** is activated a value to scale the automatic naming of the outputfiles must be entered. The shot (CMP) coordinate will be multiplied with this value. The result should be in the range 0 to 999. This number and the current filename truncated to a maximum of 4 letters define the names of the output files in the directory LINEDATA.



With **from original data** activated the program automatically searches the default seismic line datapath (raw or proc, if proc is chosen you also have the possibility to set any filefilter in addition) for the shot section with the corresponding shot and receiver coordinates and uses this filename for the pickfile naming. This might be useful if you want to use these individual pickfiles within the 2D-dataanalysis again for e.g. for a check. To be considered: existing pick files will be overwritten. With the option **ignore zero traveltimes** the zero traveltimes at the shot position will be ignored and not exported.

apply xz-topography: this option allows to automatically redefine the geometries of the shots and the receivers based on topographic xz-values stored within an ASCII-file. 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.

It is assumed that the x-values of the current traveltimes do not represent the correct x-coordinates but are determined directly on the topographic interface. 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 topographic xz-values are 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.

The following assumptions must hold true:


1. The topography file must contain the values for x=0 (fixed point) because this value serves at the starting point for determining the corrections. All corrections are relative to the value at x=0.
2. The profiledirection is pure x, this means that the y-coordinates of all shots and receivers are equal.
3. The original x-values are taken directly on the topographic interface.

The application of this option is useful if you want to introduce a topography for the wavefront-inversion or for the raytracing (chap. 4.7).

Correct lateral offset: The option allows to correct a lateral offset between shot and receivers. By default only the x-positions will be taken into account both for the shot and the receivers. Using this option corrects the x-positions of the receivers depending on the lateral offsets stored within the y-coordinate of the shot and the receivers. For this purpose the correct distance between shot and receiver will be calculated and stored on the x-coordinate of the receiver. The y-coordinates will be set to 0 afterwards.

project on x-coord.: The option calculates the distances from a given starting point for the shot and receiver coordinates and projects these distances onto the x-axis: the x- shot and receiver coordinates are set to these distances and the y-coordinates are set to 0. The option might be useful if GPS-coordinates are present as Reflexw needs pure x-coordinates for the seismic refraction interpretation. The **starting point** for the acquisition line must be entered manually. It is possible to choose between first receiver, min. x coordinate, min. y coordinates, max. x coordinate and max. y coordinate. As the loaded data have been automatically sorted with ascending coordinates the option **reload picks without sorting** allows to use the original sorting of the data (important if the starting point first receiver has been chosen). A check of the pathway can be done using the option showxyprojection under view.

As a result the x-coordinates contain the distances along the acquisition line starting at the **entered start distance** for the chosen starting point. With the option **create dst file** an ASCII dst file will be created which contains the original xy GPS coordinates together with the calculated distances along the line. This file can be used after the refraction inversion in order to reconstruct the original GPS coordinates for the inversion result (done within the traceheader menu using the type ASCII file/interpol. with activated option distances instead of tracenos). This result must represent a 2D-Reflexw file, e.g. a tomographic inversion file or when using the option fill within the modelling menu.

highlighted shot: will be used for choosing a shot number to be highlighted. The traveltimes which correspond to this shot number will be highlighted and only these traveltimes may be assigned or edited (for example remove a traveltimes value). The option might be useful for a large number of shots for which the traveltimes shall be assigned to specific layers. The speedbutton  resets the highlighted shot to 0 which means that the traveltimes for all shots may be available for assignment or editing.

5.2 traveltimes assign

After having put together the traveltimes from the several shots you may assign the traveltimes to layers. The assignment of traveltimes to layers and the subsequent combination to one forward and reverse traveltime curve (traveltimes combine, chap. 5.3) is required for the wavefront-inversion 2D (chap. 4.7.2). Forward traveltime pick means that the profile distance of the geophone location is greater than the profile distance of the shot location. Reverse traveltime pick means that the profile distance of the geophone location is smaller than the profile distance of the shot location.

The current layer number is entered within the **layer nr.** text field. If traveltimes are already assigned to layers all traveltimes corresponding to a layer are shown in the same color.

It is possible to assign a special traveltime pick to the current layer (activated option **set**) or to remove the assignment (activated option **remove**). Already existing assignments are replaced by the new ones. Therefore it is not necessary to remove a wrong existing assignment first. With the option **remove whole shot** activated all assigned traveltimes of the actual shot will be reset.

It is not only possible to assign one single pick but a total set of picks for one traveltime branch. For that purpose move along the traveltime branch with pressed mouse button and leave the button when the last wanted pick is reached.

The option **interpolate** allows to automatically assign all traveltime picks between the last two manually assigned single picks. This enables you a fast assignment of many traveltime picks of one shot. To be considered: The two picks must belong to one shot and should only belong to the forward or reverse traveltime branch.

The option **all unassigned** assigns all free picks to the current layer number.

The option **change layer nr.** allows to change the layer number of the actually assigned picks. This might be useful if you want to assign picks to an intermediate layer after having already assigned the picks to the lower one.


The option **reset** allows to reset all picks assigned to the actual layer number. With the option **all** activated all assigned picks will be set.

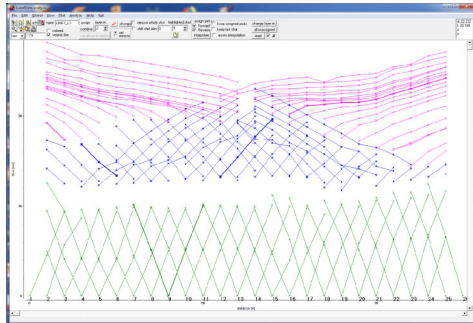
The options **Forward** and **Reverse** enable to or disable the assignment possibility to the forward and/or reverse traveltime branch. Example: With activated option **Forward** and deactivated option **Reverse** only forward traveltimes picks may be assigned.

With the option **keep last shot** activated only the picks belonging to the shot of the last pick are considered. Activate this option for example before defining the second pick for a subsequent interpolation. Then it is ensured that the interpolation is only done for picks belonging to one single shot.

With the option **keep assigned picks** activated only the non assigned picks are taken into account.

highlighted shot: will be used for choosing a shot number to be highlighted. The traveltimes which correspond to this shot number will be highlighted and only these traveltimes may be assigned or edited (for example remove a traveltime value). The option might be useful for a large number of shots for which the traveltimes shall be assigned to specific layers. The

speedbutton  resets the highlighted shot to 0 which means that the traveltimes for all shots may be available for assignment or editing.



5.3 traveltimes combine

5.3.1 combine traveltimes for the uppermost layer

Layer number 1 means that the assigned traveltimes are connected to the uppermost layer. Therefore these traveltimelpoints only contain informations about the velocities of the uppermost layer and no information about the depths. Activating the option **wavefront-inversion** applies a linear regression on all forward and reverse traveltimes between the chosen forward and reverse shot numbers. With the option **smooth inversion** activated the traveltimes will be fitted by a start velocity and a vertical gradient (useful if the traveltime curves show a significant curvature). In both cases then a completely new model is created with an uppermost layer containing the calculated velocities (and gradients respectively) at the given positions. The positions are always calculated from the mean of the individual forward or reverse pick positions stemming from one single shot.

Topography can be directly included during the wavefrontinversion for the uppermost layer 1. Activating the option topography within the CombinePanelLayer1 panel allows to automatically read the topographic values stored with the actual picks (this also automatically activates the topography option within the modelling menu, see chap. 4.4.6). These topographic pick values may come from the original seismic data or after having applied the option apply xz-topography under edit. The option altitude controls if the topographic values are depth values (option deactivated) or altitude values (option activated). The reference level defines the reference level for the altitudes. The depth axis is 0 at the reference level (see also option reference level within the model menu) - therefore the reference level should be set at least to the maximum existing altitude value. The depth values are calculated from reference level - altitude values. The program automatically calculates the distance along the topography and performs the linear regression for the velocity determination based on these distances. This is necessary as the velocity is only correctly defined if plotted against the distance along the topography. The option average topography allows to define an average range for the topography in order to remove single steps between successive topographic datapoints.


To be considered if the shot elevations are different from the receiver or surface elevations (e.g. shot within a borehole). In this case it might be useful to activate the option shift shots. The entered mean shift value will be added in order to compensate the different elevation levels.


5.3.2 combine traveltimes for deeper layers

In order to use the wavefront-inversion 2D (chap. 4.7.2) for the inversion of the traveltimelpicks assigned to one deeper special layer the individual traveltimelpicks must be combined to one complete forward and one complete reverse traveltime branch.

First you must define the **forward shot** number and the **reverse shot** number. The forward and reverse traveltimelpicks of these two shot number are the basis

for the building of the two complete traveltime branches. Therefore forward or reverse traveltime picks respectively must have been assigned to the current layer for both shot numbers before combining the traveltimes.

The combined traveltimes can be saved on disk using the speedbutton  within the combine panel. By default the filename is equal layer+current layer number.

Activating the speedbutton  within the combine panel allows you to load a previously saved traveltime curve.

There are two possibilities to combine the traveltime picks. The assigned forward and reverse traveltimes may be combined automatically (option auto combine) or manually (option manual combine).

With the option **auto combine** activated the option **generate** allows to automatically create the two traveltime branches. The forward and reverse traveltime picks of the forward shot and the reverse shot are the basis for the building of the two complete traveltime branches. These traveltimes are not changed.

Forward traveltimes belonging to shot numbers smaller than the forward shot number are combined at the beginning of the forward shot curve. Forward traveltimes belonging to shot numbers greater than the forward shot number are combined at the end of the forward shot curve. The opposite holds true for the reverse traveltimes.

If there are several traveltimes for one coordinate the automatic algorithm always uses the traveltime of the smallest shot number for the forward traveltimes and the traveltime of the highest shot number for the reverse traveltimes. If necessary the combined traveltimes are extrapolated to the smallest and greatest coordinates of the profile. The combined forward (black) and reverse (fuchsia) traveltimes are displayed after having pressed the button generate together with the maximum of the forward and reverse traveltimes and the difference in the left upper corner. The difference of the maximum forward and reverse traveltime should be small. Otherwise the assignment of traveltimes or the traveltimes itself might be in error and the picks should be reassigned (see traveltimes assign, chap. 5.2). In the combined traveltimes the redundancy of traveltimes is not considered at all. This is only considered when using the manual combine method.

The option **balance** allows you to balance these two values. In any case the wavefront-inversion 2D (chap. 4.7.2) started later uses the arithmetic mean of these two values.

With the option **manual combine** activated the assigned traveltimes are manually shifted and combined. The option **get all** extracts all assigned traveltimes between the entered forward shot number and reverse shot number. The forward traveltimes are shown as black rectangles, the reverse traveltimes are shown as pink rectangles. In addition the traveltimes belonging to the chosen forward and reverse shot number are plotted bold. With activated option show original the original data are plotted in addition.

It is possible to move in time a complete group of forward or reverse traveltimes belonging to one shot. Click on one traveltime of that group you want to move and move the mouse with pressed button. The traveltimes belonging to the chosen forward and reverse shot number cannot be moved.

After having moved the traveltimes to the wanted positions it is still necessary to generate a complete forward and reverse traveltimes curve. This means that the maximum forward and reverse traveltimes must be determined if not existing and double datapoints must be smoothed. Both is done using the option **generate**. The option **medianfit** controls if the smoothing is done using the mean (deactivated) or the median (activated). This means that data redundancy is considered in the definition of the combined traveltimes.

To be considered: In both cases (automatic and manual combine using the option generate) the forward and reverse traveltimes branches are automatically extrapolated both to the forward and reverse distance if no traveltimes picks exist at those positions.

Activating the option **wavefront-inversion** opens the menu modelling for a subsequent wavefront-inversion 2D (chap. 4.7.2). First you must load the wanted overburden model. Then you have to enter the wanted space increment DeltaX and then you can start the wavefront-inversion.

5.4 time term analysis

The timeterm analysis (option Time Term interpretation under Analyse) allows the reconstruction of a 3D-refractor from xy-traveltimedata. The timeterm analysis has been described by Willmore and Bancroft (1960). The essential feature of the method is that each traveltime t_{ij} may be written in the form $t_{ij} = a_i + b_j + \Delta_{ij}/v$, where a_i and b_j are timeterms which are characteristic of the shot- and receiver point respectively, Δ_{ij} is the distance between shot and receiver and v is the refractor velocity. Under special conditions it is possible to derive a_i , b_j and v which give the best fit to the observed traveltimes t_{ij} . The preconditions are:

- the velocity of the overburden varies only with depth within the critical refracted ray cone under the shot or receiver
- the refractor velocity is assumed to be constant
- slope and curvature of the refractor is small
- the model consists of one layer and a half space

If the current layernumber (option layer nr.) is set to 0 all traveltime data above the min. traveltime are taken into account. If the current layernumber is unequal 0 only the traveltimes which have been assigned to that layer before are taken into account. After having activated the option Time Term interpretation under Analyse the wanted traveltime data are queried. The time term analysis window opens. The following options are available:

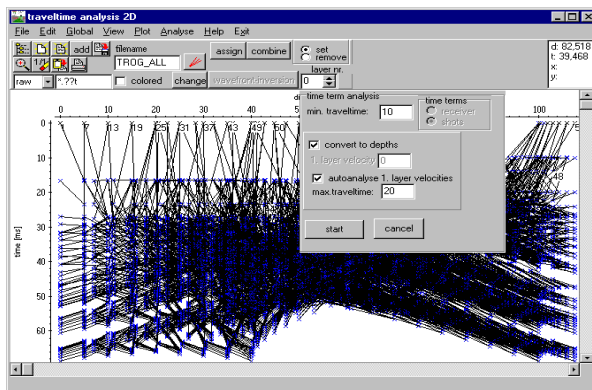
min. traveltime: specifies the lower border for the traveltimes which are taken into account for the timeterm analysis (layer nr. = 0).

Within the time terms box you may specify if the timeterms are calculated for the different receivers or shots. With the option autoanalyse 1. layer velocities activated the timeterms are always calculated for the different shots.

convert to depths: if activated the depths are calculated in addition. For that purpose the velocity of the overburden must be entered (option 1. layer velocity).

With the option

autoanalyse 1. layer velocities activated the velocities for the overburden are calculated from the given traveltimes. The option max. traveltime specifies the upper border for the traveltimes which are taken into account for this calculation.



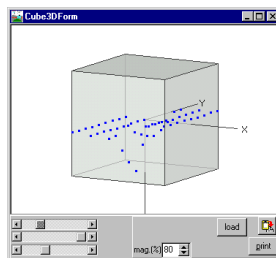
After having activated the option start the traveltimes data are sorted and analyzed. The result is plotted into the 3D-cube and is saved under an ASCII-file. The output is an ASCII-file with the first line showing the velocity of the refractor. The following lines contain the resulting time terms and the coordinates:

timeterm x-coord. y-coord. overburdenvelocity depth

```

output on file TRDG_ALL.TimeTerm
2117.944655
10.57 0.00 -20.00 0.00 0.00
10.83 20.00 -20.00 300.62 3.26
10.98 40.00 -20.00 300.63 3.30
8.47 60.00 -20.00 500.80 4.24
8.31 80.00 -20.00 500.54 4.16
8.04 100.00 -20.00 500.72 4.02
10.56 0.00 -15.00 302.35 3.19
10.82 20.00 -15.00 302.03 3.27
10.97 40.00 -15.00 301.5 3.3
9.92 60.00 -15.00 501.2 4.04
8.28 80.00 -15.00 500.96 4.15
8.03 100.00 -15.00 501.18 4.03
10.60 0.00 -10.00 302.86 3.21
10.87 20.00 -10.00 302.88 3.29
13.65 40.00 -10.00 302.58 4.13
10.62 60.00 -10.00 500.97 5.32
8.29 80.00 -10.00 500.67 4.15

```



6. Seismic crosshole testing

The program **seismic crosshole testing** has been designed for a fast interpretation of crosshole test data.

6.1. Original data

A crosshole test requires that p- or s-wave source and receivers are placed always on the same depths/elevation during recording. In order to benefit from the polarisation characteristics of the s-wave the excitation can be performed in two directions (later called plus and minus directions). If only one direction or a p-wave source has been used the excitation option has no meaning.

The acquisition software should be able to generate one (stacked) data file per depth and excitation direction stored using the SEG-2 format.

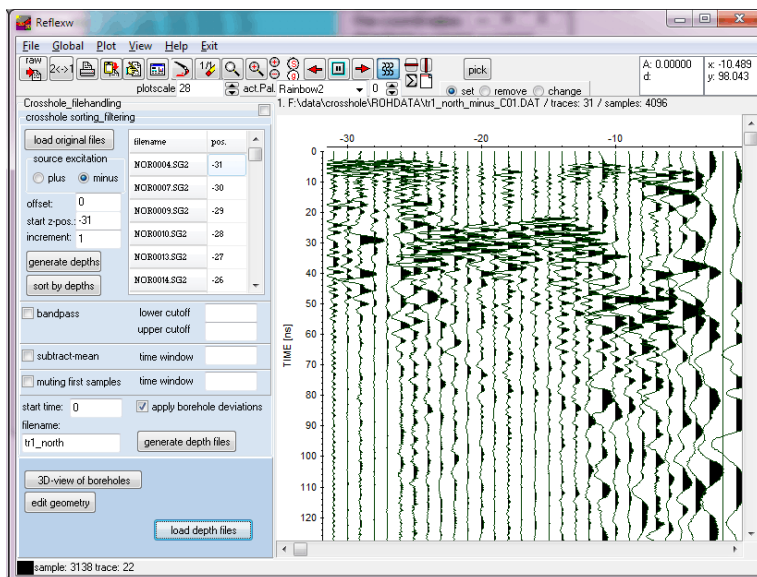
The channels of a multi-directional receiver are connected in a fixed order to the seismograph. Geophones of Geotomographie GmbH have always the following arrangement:

Channel 1 - vertical component

Channel 2 - horizontal component agreeing with the azimuth reading of integrated compass

Channel 3..n - remaining horizontal components

Collect all gathered seismic records in a single directory. Each record should contain the seismic signals recorded at a single depth, with a single excitation direction. The filenames should contain a record number and must be in SEG-2 format.



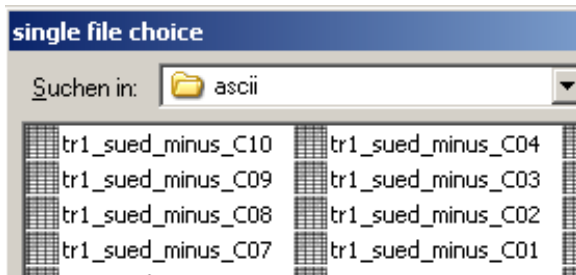
6.2. Channel sorting and filtering within seismic crosshole testing

The first step is to assign a depth for each file, slit up the seismic record sorted by channel and generate seismic files sorted by depth:

- load the original records of one excitation direction. Make use of the multiple selection feature of the dialogue window. One can select more than one file by pressing the < Control > - key together with the mouse button.
- Assign a z-position for each file, filling in the table by hand or using the standard assignment (option generate depths) with predefined default parameters (start z-pos. and increment). If a borehole deviation shall be applied afterwards the values must be negative (e.g. -2 to -20) which means that the first position is 2 m below the top of the borehole.
- Make sure that the list is ordered correctly by depth using the option sort by depths if the depths have been entered or changed manually.
- enter the correction source excitation (minus in this example) and the offset between the two boreholes (in this example 10 m). Enter the start time if a pretrigger has been used (start time must be negative for a pretrigger).
- Enter any filter option bandpass, subtract-mean or muting first samples and generate new seismic files sorted by depth using the option generate depth files based on the entered filename.
- Optionally activate the option apply borehole deviations in order to apply the borehole coordinates which must have been stored within 2 different ASCII-files. This correction can also be done afterwards (see chap. 3). Each line of the ASCII-file contains the XYZ-coordinates in meter. The Z values must be given as above sea level. The first line corresponds to the borehole top level. The original z-positions (see item 2 above) are assumed to be relative to this top level and must be negative. After having pressed the button generate depth files the program asks the two borehole filenames (multiple file choice). It is assumed that the entered z-positions have been measured along the boreholes. The program recalculates these positions both for the shots and the receivers based on the given borehole data. A linear interpolation is done between the individual borehole coordinates.

filename	pos.
NOR0004.SG2	-31
NOR0007.SG2	-30
NOR0009.SG2	-29
NOR0010.SG2	-28
NOR0013.SG2	-27
NOR0014.SG2	-26

Using the option **generate depth files** for each excitation direction and each receiver channel (corresponds to the component) a new Reflexw file has been generated. The channel number and the excitation direction are part of the new filename, for instance: tr1_sued_minus_C05.dat



The data will be stored under the directory ROHDATA under the actual project directory.

The option load depth files allows to load up to 4 different files using the multiple file choice.

6.3. Apply borehole deviations and edit geometry

The option edit geometry within the crosshole_filehandling panel allows to edit the shot and receiver coordinates of the actually loaded depth file, e.g. apply the borehole deviations.

After having activated the option the menu edit trace header coordinates opens together with a table containing the traceheader coordinates and the traceheader gain. The table shows the traceheader coordinates (distance and the x, y, z-coordinates of the receivers and the shots), the time delay, the gain and the time collect. These values can be manually changed. The following options are available:

The option **apply borehole deviations** allows to apply the borehole coordinates which must have been stored within 2 different ASCII-files. Each line of the ASCII-file contains the XYZ-coordinates in meter. The Z values must be given as above sea level. The first line corresponds to the borehole top level. The original z-positions (see also chap. 6.2) are assumed to be relative to this top level and must be negative. The original xy shot and receiver coordinates will be ignored. After having pressed the button generate depth files the program asks the two borehole filenames (multiple filechoice). It is assumed that the original z-positions have been measured along the boreholes. The program recalculates these positions both for the shots (corresponds to borehole 1) and the receivers (corresponds to borehole 2) based on the given borehole data. A linear interpolation is done between the individual borehole coordinates. The following table shows the file structure of 2 borehole data and the pictures show the coordinates before and after the borehole correction. To be considered: The first receiver within the example corresponds to the greatest depth.

50000.0 45300.0 110.0
 50000.1 45300.1 108.5
 50000.2 45300.2 106.5
 50000.3 45300.2 102.5
 50000.4 45300.1 100.5
 50000.3 45300.0 95.5
 50000.2 45300.1 90.5
 50000.1 45300.2 85.5
 50000.1 45300.3 80.5

50005.0 45304.0 110.0
 50005.1 45304.1 106.5
 50005.0 45304.2 102.5
 50004.9 45304.2 100.5
 50004.7 45304.1 98.5
 50004.4 45304.0 92.5
 50004.2 45304.1 88.5
 50004.1 45304.2 84.5
 50004.1 45304.3 81.5

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	0	0	0	-31	10	0	-31	0	1	0
2	0	0	0	-30	10	0	-30	0	1	0
3	0	0	0	-29	10	0	-29	0	1	0
4	0	0	0	-28	10	0	-28	0	1	0
5	0	0	0							
6	0	0	0							
7	0	0	0							
8	0	0	0							
9	0	0	0							
10	0	0	0							
11	0	0	0							
12	0	0	0							

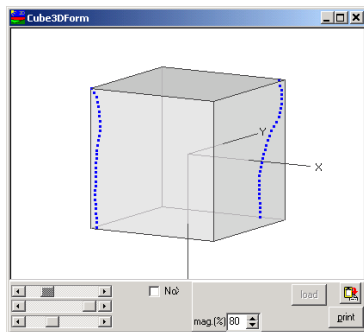
CrossholeGroupBox:
 apply borehole deviations
 3D-view of boreholes
 sample number: 4096
 trace number: 31
 TimeDimension
 start time: non.frequency
 time increment: 0.03125

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	0	50000.1	45300.3	79.02488795	50004.1	45304.3	79.03905291	0	1	0
2	0	50000.1	45300.3	80.02488795	50004.1	45304.3	80.03905291	0	1	0
3	0	50000.1	45300.289504	81.024783007	50004.1	45304.3	81.03905291	0	1	0
4	0	50000.1	45300.269506	82.024583067	50004.1	45304.282041	82.03875363	0	1	0
5	0	50000.1	45300.249512	83.024383127	50004.1	45304.248726	83.03819860	0	1	0
6	0	50000.1	45300.229516	84.024183187	50004.1	45304.215411	84.037643507	0	1	0
7	0	50000.1	45300.209520	85.023983247	50004.1	45304.186572	85.037051162	0	1	0
8	0	50000.110472	45300.189526	86.023678624	50004.138410	45304.161585	86.036426747	0	1	0
9	0	50000.130465	45300.169534	87.023278864	50004.163395	45304.136604	87.035802332	0	1	0
10	0	50000.150457	45300.149542	88.022879103	50004.168375	45304.111620	88.03517791	0	1	0
11	0	50000.170445	45300.129590	89.022479343	50004.226702	45304.086646	89.034053687	0	1	0
12	0	50000.190441	45300.109596	90.022079583	50004.276624	45304.061687	90.032494830	0	1	0

CrossholeGroupBox:
 apply borehole deviations
 3D-view of boreholes
 sample number: 4096
 trace number: 31
 TimeDimension
 start time: non.frequency
 time increment: 0.03125
 source <-> rec.
 check rec. coordinates
 factor f check: 10
 smooth rec. xy: coord.
 factor f: smooth: 4
 interpolate
 load from AsciiFile
 save on AsciiFile
 reload from file
 save changes
 close

The option **3D-view of boreholes** allows a 3D-view of the shot and receiver xyz-coordinates. The cube may be interactively rotated using the left mouse key.



nominal frequency: enter the nominal frequency of the antenna (GPR-data) or seismic source (seismic data) used. Enter the value in MHz with timedimension set to ns, KHz with timedimension set to μ s or Hz with timedimension set to ms. The option get allows to automatically determine the nominal frequency from the data.

start time: specification of the start time for sample 1 in the given time dimension.

TimeDimension: Specification of the time dimension. Possible inputs are ms and ns.

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

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

The option **save on AsciiFile** stores the traceheader coordinates on an ASCII-file with the extension dst. Each line of the ASCII file contains the following 8 informations:

tracenumber distance Shot-X-Pos Shot-Y-Pos receiver-X-Pos receiver-Y-Pos Shot Z receiver Z

The option **load from AsciiFile** allows to load the traceheader coordinates from an ASCII-file with the extension dst. Each line of the ASCII file contains the following 8 informations:

tracenumber distance Shot-X-Pos Shot-Y-Pos receiver-X-Pos receiver-Y-Pos Shot Z receiver Z

An automatic interpolation is done if for various traces no coordinate data are present within the ASCII-file.

If all distances within the ASCII-file are zero (2.column) the distances will be calculated from the total difference between each shot and receiver positions (x,y and z-coordinate are taken into account).

The option **interpolate** allows to interpolate between two given coordinates. The interpolation is only possible for one single traceheader coordinate group (for example only rec-z). There are two ways how to proceed:

a. activate the option interpolate, then click on the wanted first coordinate and then on the second one. Inbetween a linear interpolation will be done.

b. If you have changed manually any coordinate click on interpolate and then on the second coordinate. Again the linear interpolation will be performed.

The option **source <-> rec.** allows to exchange the shot and receiver coordinates. The option might be useful if the shot position has been continuously changed with a fixed receiver. Such a geometry should be exchanged because the following interpretation methods all base on a fix shot position and changing receiver positions.

The option **x <-> y** allows to exchange the x- and y-coordinates of the sources and the receivers.

The option **check rec.coordinates** checks the receiver coordinates for breakouts and interpolates these. For that purpose the two x and y mean absolute differences between successive xy-receiver positions are calculated. If the absolute difference between two successive xy-receivers positions multiplied by the value of the option factor f.check exceeds one of the two mean difference value an interpolation for both the xy-shot and receiver coordinates is done for this xy-position.

The option **smooth rec. xy-coord.** allows to perform a running smooth of the xy-receiver coordinates over an adjustable range of traces. The option might be useful for GPS-data especially if the option update distances (see below) shall be used afterwards.

6.4. Arrival time picking

The last step consists in the picking of the first arrivals.

The Reflexw files which are sorted by depths are loaded using the option load depth files. Up to 4 different files can be loaded using the multiple file choice.

Theoretical Considerations

Primary (P-) and Secondary (S-) waves are different types of body waves. The p-wave is in principle at least 1.4 times faster than the s-wave. The ratio can go up to about 7 or even 12 in case of loose saturated sediments.

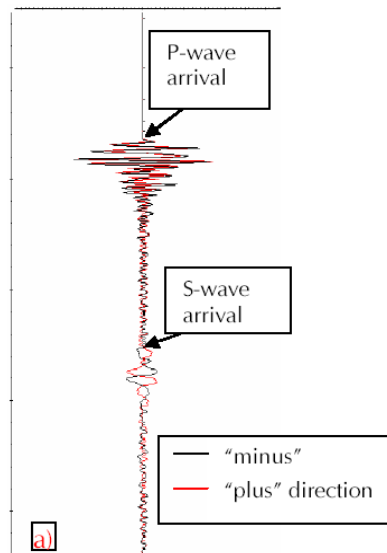
The direction of the particle motion caused by a p-wave (as a compressional wave) does in general not depend on the direction of the excitation whereas the direction of particle motion caused by a swave does. Therefore if the signals of the two opposite excitation directions are overlaid the wavelets of the p-wave will show the same polarization direction whereas the signals at the time of s-wave arrival will split up leading to a phase inversion between the two s-wave wavelets.

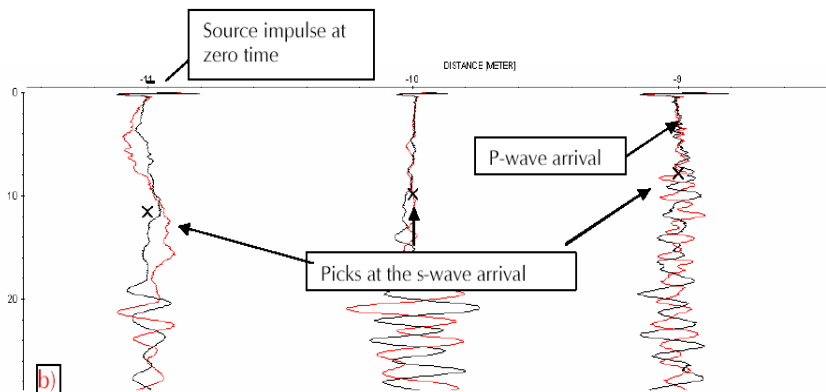
Figure a) gives an example of seismic signals of an extraordinary good quality. These signals were recorded in clay.

The figure shows also that the wavelet of the p-wave has a high frequency in comparison to the s-wave wavelet.

Figure b) shows that the electrical impulse used to drive the borehole source can be seen in the recorded signals under certain conditions. This is caused by cable cross-talk and cannot be eliminated completely. Please don't mistake this pulse for the p-wave arrival.

Due to the nature of the crosshole test only the horizontal channels of the geophone are relevant, both for p- and s-wave arrival identification. Since the orientation of the geophone is not controlled during the test it is in the beginning unknown which of the horizontal channels is most appropriate for the arrival time picking. Therefore it is good practice to select at first a channel where most signals have a reasonable data quality, perform the picking and adjust the picks afterwards using the remaining horizontal channels.



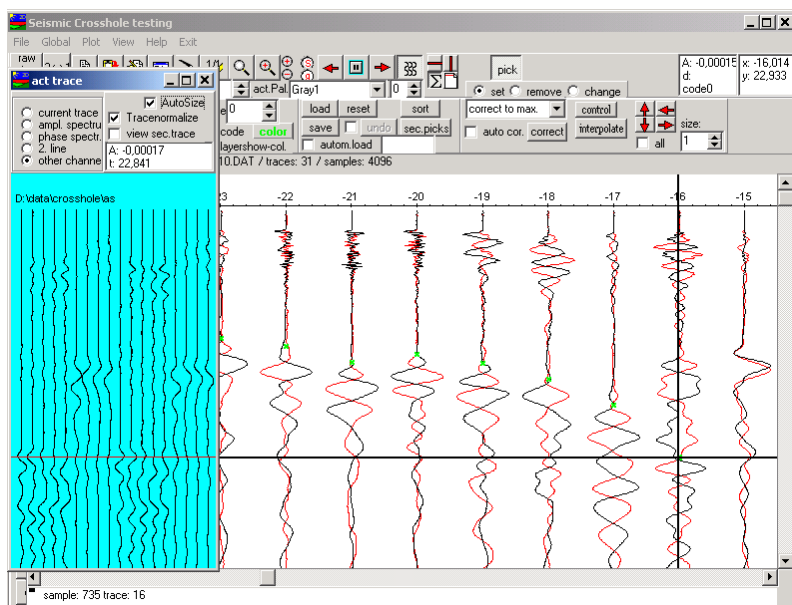
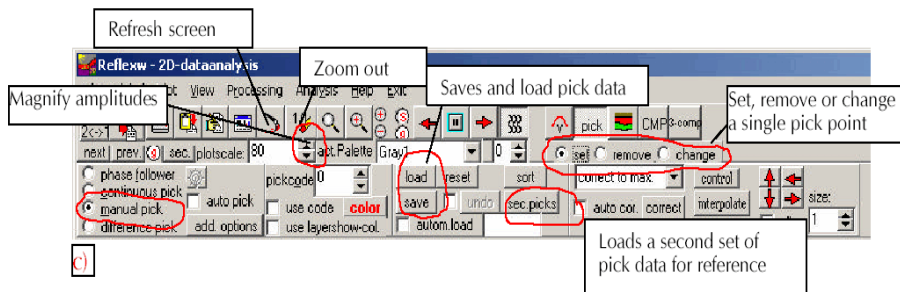


Technical Considerations

Technically the arrival picking means to set one pick point indicated as “x” to each signal. In the beginning only the buttons and control elements indicated in Figure c) are needed. Manual picking is in almost all cases required, since the automated algorithm (“phase follower” control) does not work very well on crosshole data. Please experiment yourself, most is self-explanatory. You can call the help system of seismic crosshole testing for additional advice by pressing the “F1” key.

But please consider the following:

- Pick P-wave and S-wave arrival separately.
- Be careful not to set more than one pick on each trace.
- If you are unsure about where to set a pick on a trace just leave it blank. It is usual in crosshole testing that at certain depths no reliable velocity can be determined.
- Make use of the “sec.picks” button if you are unsure about the arrival times. If you save different picking versions you can use this button to load an alternative picking scenario in the background for comparison.
- Optionally you can use the view/wiggle window with activated option **other channels** which allows the plotting of the actual trace (depth) of all other channels within the actual project. The program searches all files within the actual directory which has the same filename except the excitation direction (plus or minus) and the channel number (C??). The current trace of all these files will be plotted within the wiggle window together with a straight line indicating the timebase of the cursor position. This option may help to determine the correct onset if it is better visible in any other channel.
- Save the picks occasionally and at the end of picking according to the procedure described later.



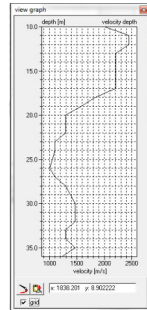
Saving of pick data

Push save button to save the pick data set.

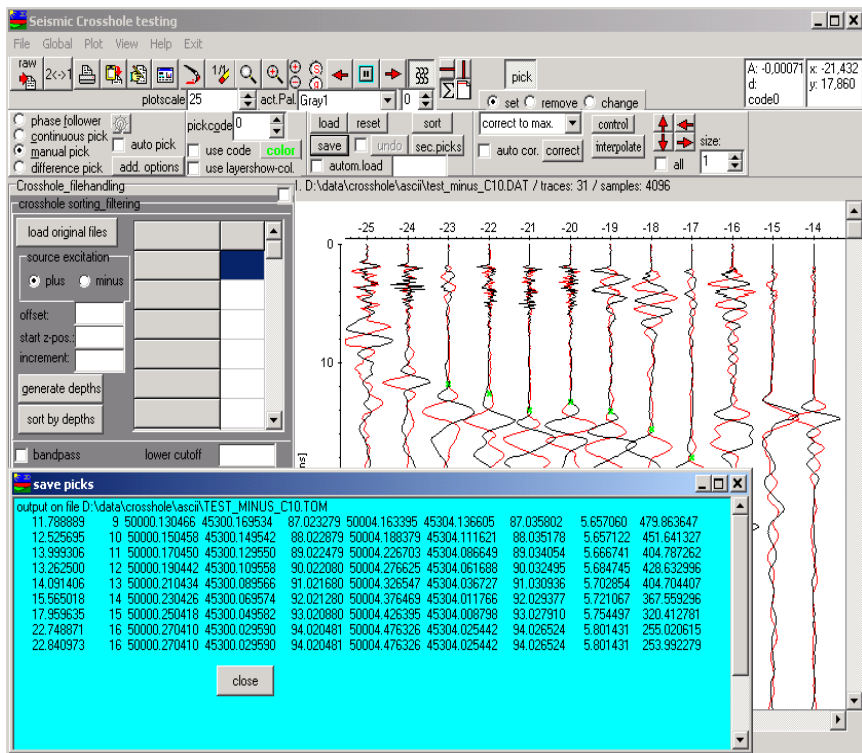
Chose a filename for each set, e.g. "p-wave" or "s-wave".

Choose one of the following formats:

ASCII-3D tomography: this is the standard ASCII-format. The ASCII-file contains the traveltime, the tracenummer and the XYZ-coordinates of the shot and the receiver as well as the distance between shot and receiver distance and the velocity (last column) calculated from this distance and the traveltime (see picture below). With the option **view graph** activated a velocity depth (z shot coordinate) will be displayed. This graph can be copied to the clipboard.



Reflex Win: binary format for the interchange with the Reflexw program format and if the option sec.picks is used.



Literature

Aldridge, D.F., and Oldenburg, D.W.: Refractor imaging using an automated wavefront reconstruction method: *Geophysics*, v.57, p.378-385, 1992.

Al-Chalabi, M. 1974, An analysis of stacking, RMS, average and interval velocities over a horizontally layered ground: *Geophysical prospecting*, 458-475

Bilas, E. 2007, Long-Offset Approximations to NMO Function for a Layered Subsurface: 69th EAGE Conference and Exhibition incorporating SPE EUROPEC 2007

Booth, A.D., Clark R. and Murray, T.: Semblance response to a ground-penetrating radar wavelet and resulting errors in velocity analysis, *Near Surface Geophysics*, v.8, p.235-246, 2010.

Daniels, D.J. Ground Penetrating Radar 2nd Edition, IEE Radar, Sonar, Navigation And Avionics Series 15, ISBN 0 86341 360 9, pp. 726, 2007.

Gilbert, P., Iterative methods for the three-dimensional reconstruction of an object from projections. *J. Theor. Biol.*, 36, pp. 105-117., 1972.

Hatton, L., Worthington, M.H., & Makin, J., *Seismic data processing: Theory and practice*. Blackwell Scientific Publications, 177 pp., 1986.

Herman, G. T., *Image reconstruction from projections - The fundamentals of computerized tomography*, Academic Press, New York, 316 pp., 1980.

Kelly, et. al., synthetic seismograms: a finite difference approach, *geophysics*, 41, 2-27, 1976.

K. Knödel, H. Krummel und G. Lange, *Handbuch zur Erkundung des Untergrundes von Deponien und Altlasten*, Band 3 Geophysik, 2. Auflage, Springer Verlag, 2005, ISBN 3-540-222758

Nolet, G., *Seismic tomography*, D. Reidel Publ. Co., Dordrecht, pp. 386, 1987.

Pratt, W.K. *Digital Image Processing*, 2nd. ed. John Wiley & Sons, New York, 1991, 248, 503

Reynolds, boundary conditions for the numerical solution of wave propagation problems, *geophysics*, 43, 1099-1110, 1978.

Sandmeier, K.J. and F. Wenzel, Synthetic seismograms for a complex crustal model. *Geophysical Research Letters* 13, 1986.

Stolt, R.H., Migration by Fourier transforming, *Geophysics* 43, 23-48, 1978.

Thornburgh, H.R.: Wave-front diagrams in seismics interpretation, *AAPG Bull.*, 14, 185-200, 1930.

Vidale, J.: Finite-Difference calculation of travel times, Bull. seism. Soc. Am., 78, 2062-2076, 1988.

Whiteley, R.J., Holmes W.H. and R.D. Dowle, 1990. A New Method of Downhole-Crosshole Seismics for Geotechnical Investigation, Exploration Geophysics, 21, 83-89.

Willmore, P.L. and A.M. Bancroft, 1960. The time-term approach to refraction seismology, Geophys.J., 3, 419-43. In order to use the wavefront-inversion 2D (chap. 4.7.2) for the inversion of the

Worthington, M. H., An introduction to geophysical tomography, First Break, 20-26, 1984.

Yilmaz, Özdoğan : Seismic Data Processing, Investigations in Geophysics, Society of Exploration Geophysicists, 526 pp., 1987, ISBN 0-931830-40-0

Index

1D models	187, 215, 650
1D-Filter	331, 333-338
1D-models	645, 647
2D interval velocities	
CMP 2D model	550
2D vrms velocities	
CMP 2D-model	549
2D-Data-Analysis	193
2D-Filter	406, 408-413
3-component analysis	526
3D diffraction stack	434
3D Kirchhoff migration	435-437
3D picking	607
3D-cube	588
3D-cube display	588
3D-data, import	253
3D-datafile	593
3D-datainterpretation	555
3D-FD	668
3D-FD	663
3D-File Menuitem	559, 616, 619-621, 623, 624
3D-file, import	254
3D-semblance	438
3D-topography	363
Act. trace, display	214
Add random noise	377
add. 2column data	
view	221
AGC-Gain	343
AGC-Gain, display	276
AGCGain	
plotoptions	276
Air layer, correction	362
altitude	651
amplitude calibration	
layer velocity determination	502
Amplitude, instantaneous	375
Amplitudes, display	214
Analysis Menuitem	461
API key	209
Arithmetic functions	371, 372
ASCII	251
ASCII-MATRIX	251
ASCII-pick difference	
save picks	485
assign	
traveltimes	688
auto correlation	
1D-filter	339
Autointerpolation, display	277

autom.load	470
automatic picking	468
autostart	468
autostop	468
average layer	
model generation	657
Average xy-filter	412
Averaging	333
Axis, labeling	277
B-Scan	193
Background removal	409
Background removal, sliding	408, 635
Bandpassbutterworth	335
Bandpassfrequency	334
Banner paper, print	285
Batch printing	282
batch processing	
sequence processing	459
BatchStartFD	665
BGREMR	245
bitmap	
interactive choice	204, 217
print preview	287
Bitmap size[kb]	209
Bitmap-Import	233
C-scan	556, 598
calc. traveltimes	646
calculate traveltimes differences	640, 689
calculated traveltimes	646
change pick code	
picking	476, 477
Check disk space	208
check source/receivers positions	486
Clipboard	202
CMP 2D-model	549
CMP semblance analysis	546
CMP sort	692
CMP velocity adaptation	542
CMP velocity analysis	532
CMP-bin	516
CMP-data	532
CMP-processing	508, 521, 525, 529
CMP-processing geometry	511
CMP-processing stack	517
CMP-velocity parameters	211
ColorBars, display	214
Combine files f.CMP	401
combine layers	
model generation	658
comment boxes	
print preview	286
comment markers	521
Comment, fileheader	291

Comment, print	283
Commentmarker:	278
compensate stripes	348
complete plot by scrolling	208
Complex trace-analysis	375
Complex trace-analysis/spectral analysis	373
Compress 3D-file	412
Compress data	410
constrain borders	414
construction change	209
layer show velocities	503
pick	468
contour lines	268
control xy-coord	
picking	210
ConversionMode	253
ConversionMode, sequence	253
coordinate transformation	
traceheader tabella	321
Coordinates, file	288, 289, 291-293
Coordinates, project	324
Coordinates, traces	293
Copy, file	202
Core	463
Core data	215, 650
core data/1D-models	
layer velocity determination	502
core parameters	210
Corefile	549, 550
Corr.max. phase/wrap	360
Correct 3D-topography	363
correct elevations	
plot options	280
Correct for 2 layers	362, 363, 423, 433
Correct max. phase	359
Correct phase	359, 361
Correct picked phase	361
create 2D-model	647
create MPEG-file	
3d-datainterpretation	568
Crosscorrelation	338
crosshole tomography	669
crossing line	
picking	476
crossing line picking	476
CSV-GPS	
traceheader coordinates	297
cube	
3D-display	588
Cube, move through	573, 590
cur. picks	
edit comment marker	522
curves size	

CMP-velocity analysis	211
Damping, compensation	344
Data cube	555
data fusion	446
data traveltimes	645
data type:	238
Data-interpretation	555
Decimal separator	29
Declipping	370
Declipping/arithmetic functions	368
Declipping/max.value	370
Declipping/plateau	370
Declipping/threshold	370
Deconvolution	451
Deconvolution/shap.	451
Depth-axis, show	277
Dewow	338
plotoptions	277
difference	
picks	189, 468
difference pick	468
differentiation	
1D-filter	454
Diffraction	462
Diffraction 2D-veloc	426
Diffraction stack	417
Diffractions, contraction	417, 420, 421, 423, 434-437
digitized signal	
FD-modelling	661
Dijkstra algorithm	678
dilation/erosion	
3d-datainterpretation	572, 575, 585
Dip filter	442, 443
Directory structure/project directory	23, 25
dispersion curve	445
Display resolution	29
display scale:	
CMP-velocity analysis	211
Display, 3D-data	573, 583
distance markers	521
Distortion free	573, 584
div. compensation	
gain	345
downhole	462
Drift, correction	359
Duplicate traces	399
DXF	
picking	482
Dynamic correction	355
edit color palettes	
edit color palettes	273
Edit several FileHeaders	324
Edit traces/traceranges	393, 395-402

eikonal equation	146, 678
Elevation	
plotoptions	280
Energy decay	343
Energy decay, display	276
EnergyDecay	
plotoptions	277
Envelope	375
Expand 3D-file	
2D-filter	412
Expand data	410
Exploding-Reflector-Model, modelling	639
export calc.traveltimes	646
export data traveltimes to ASCII	646
Export Menu	262
export Reflexw rasterfiles	646
export several existing picks into 1 ASCII-file	191, 481, 483, 486
export to ASCII	
traveltimes	689
Export, data	262
Extract traces	397
extract wavelet	456
extrapolate	
modelling	637, 658
fast data cube display	605
FD-migration	428
FD-simulation, modelling	127, 637
FD-vidale	678
File MenuItem	201
File, export	262
File, filenamefactor	241
File, import	225, 226, 229, 238, 241, 243, 250, 253-256, 258, 261
File, primary	201
File, second	201
Fileheader	238, 288
Fileheader ControlPanel	292
Fileheader coordinates	288
FileHeader Edit	288
Fileheader filename specification	291
Fileheader input1	289
Fileheader time specification	289
Fileheader, coordinates	238
FileInfo, display	214
Filename, automatic	241, 291
Filename, manual	241, 291
Filename, number	240, 241
Filename, specification	241, 291
fill	
modelling	642
Filter/timedependent	336
finite difference	678
First arrival, correction	359
first arrivals	71, 129, 678, 688

Fk filter	441
Fk filter-lineparts	443
Fk migration (Stolt)	421
Fk spectrum	444
fk-filter/data fusion	439
Fk-filter/Fk-spectrum	415
Flatten, phase	359, 361
FlipProfile	386
FlipXAxis	279
flipXYsorting 3D-file 3d-datainterpretation	568
FlipYAxis	279
Fonts	208, 284
FontSettings	284
forward shot	697
free of distortion interactive choice	203, 216
free surface FD-simulation	662
Frequency analysis	375, 378
Frequency analysis, moving window	376
Frequency attenuation, compensation	453
Frequency, instantaneous	375
Gain	341, 343-347
Gain function	344
Gain, display	276
Gain, fileheader	291
Generate 3D-file from 2D-lines	593
generate batchfile batch processing	459
generate pickfile (2-way traveltimes)	642
Generate single timeslices	598
Geometric spreading, compensation	344
Geometry file, import	254, 255
Geometry, CMP-processing	511
georeferenced tfw	265
GEOSCAN32	234
GeoTiff 3D-cuts	576, 585
GeoTomoCG picking	484
Global MenuItem	208
Global settings	28, 208
GPS Radan - GSSI	229
traceheadercoordinates	296
Gps map	219, 566
Graphics resolution:	12
Grid, show	277
GSSI-GPS	301
HCD-Format	236
Hilbert-Transformation	375
histogram	222

histogram equalization	572, 575, 585
Horizontally coherent energy, emphasizing	408
Horizontally coherent energy, suppression	409
IDS-Format	232
IDS€Format	232
import	
automatic import	225
Import ControlOptions	243
Import ControlPanel	258
Import ConversionMode	253
Import Fileheader-coordinates	238
Import filename specification	241
Import format specification	60, 250, 314
Import HeaderInput	261
Import Menu	225
Import, data	225, 238, 241, 243, 250, 253, 255, 257, 258, 261
IMPULSERADAR	235
Infrasense	232
Inline lines, import	254
Input format	225, 250
Insert profile	400
Insert zero traces	404
Integration	454
Interactive choice	203
intercept time	321, 462, 544
interpolate actual 3D-file	
3d-datainterpretation	568
Interpolate non-equidistant data	384, 385
Interpolate traces	396
interpolated physical parameters	649
interpolation	
traceheader coordinates	295
Interpolation, data	381
Isolines	268
key shortcuts	200
Kirchhoff 2D-veloc	427
Kirchhoff migration	419
Language	208
last pick stop	468
latency	307
layer thickness	
picking	476
save picks	485
layer velocities	
layershow	501
layer velocity determination	502
Layer, pick	466
LayerShow amplitude, calibration	504
LayerShow MenuItem	491
LayerShow multioffset picks	506
LayerShow parameters	211
LayerShow report, create	497
LayerShow velocities, create	501

LayerShow, create	494
legend	
layershow	492
Linear features	462
load perpendicular picks	
picking	476
Low amplitude, emphasizing	343, 344
Lowpass filtering	333
Make equidist.traces	384
MALA RD3	226, 228
MALA RD6	228
Manual gain (x), changing of	346
Manual gain (y), changing of	345
Manual scaling	276
Map	203
Marker, show	277
Markerinterpol	381
MarkerRelocate	292, 329
markers	521
mean velocities	
layershow	501
Mean velocity	536
meandering	
data-import	256
meandering 2D-profiles	59
Meandering data	391
Meanfilter	333
median filter	
3D-datainterpretation	572, 575, 585
Median xy-filter	413
Medianfilter	333
Merge files	402
merge in timedir	404
Migration	417, 419, 421, 426-428, 431, 434-437
topographic	422
Mix profile	449
Model contours	564
Model generation	651
Model parameters, input	652
Model simulation	660
Model, load	644
Model, save	644
Modellayers, show	649
Modelling	637
Modelling File MenuItem	644, 647
Modelling View MenuItem	649
Modelling, DoRaster	663
Modelling, Exploding-Reflector-Model	639
Modelling, FD-computation	660
Modelling, FD-computation of several single lines	666
Modelling, plane wave	638
Modelling, point-source	639
Modelling, setting of FD-parameters	660

Modelling, space-increment	638
Modelling, StartFD-computation	664
Modelling, time-increment	638
Modelling, tomography	669
Modelling, Tomography, check rays	675
Modelling, Tomography, DoRaster	675
Modelling, Tomography, SIRT	669
Modelling, Tomography, stopping criteria	670
Modelling, Tomography, travel time data	673
Modelling, topography	659
modelparameters	
show	649
Modelpoints, show	649
move picks	472
Move starttime	355
Move traces	400
Moveout-data	532
Moving window spectrum	376
Multichannel	
import	242
multichannel data	404
multioffset data	
layer-show velocities	501, 506
multioffset picks	
layer velocity determination	502
multiplexed data	
import	242
multiplexing	404
Muting	356
national conversion	313
network raytracing	678
NMO, CMP-processing	517
NMO-const.vel.	
stack	517
NMO-correction	
CMP-processing	520
Noise suppression	334-337, 408
Noise suppression, dip	442, 443
Non-equidistant, data, show	277
Notchfilter/frequ.	337
Offset-bin	516
Open 1.-4. Line	
2D-dataanalysis	201
Open file	201
Output format	250
output type	
FD-modelling	662
overlay	
3D-data	557, 571
plotoptions	278
Parallel lines, import	254
Phase follower	189, 466
Phase, correct	359, 361

Phase, flatten	359, 361
Phase, instantaneous	375
pick	
3D-datainterpretation	607
Pick MenuItem	466
pick parameters	210
Pick save MenuItem	480, 487
Picking, display	210
PickPanel, display	214
Picks	
3d-datainterpretation	563
Picks for 3D-file	563
PixelsPerSample	266
PixelsPerTrace	266
Plane wave, modelling	638
Plot	213, 266, 268, 270, 271, 275-277
Plot MenuItem	213
Plot, change color	270, 275
Plot, marker	277
Plot, palette	270
Plot, pointmode	270
Plot, wigglemode	275
PlotGain	276
PlotOptions	266
Plotsettings	266
Plotsuboptions	277
Podvin	678
Point-source, modelling	639
Pointmode	266
Pointmodeattributes	270
prestack migration	
migration	431
Print	208
3D-datainterpretaton	560
print header	286
Print Menu	282
Print preview	286
Print, comment	283
Print, file	282-284
Print, several files	284
PrinterSetup	208
PrinterSize	283
Printing on banner paper	285
PrintOptions1	282
PrintOptions2	283
Proceq	
seggy-import	244
Processing	
3d-datainterpretation	556
Processing flow, file	202
Processing flow, sequence proc.	332, 457
Processing MenuItem	330
ProcessingLabel	351

profile line (trace header coord.):	216
Profile map	203
profile xy-location.	216
Project directory.	23, 25, 201
PULSEKKO.	226, 243
RADAN	228, 244
RAMAC	243
RAMAC-borehole.	296
RAMAC-GPS.	296
random noise suppression	442
random perturbation	
modelling.	655
raytracing.	677
RDEconversion (netherlands)	313
receiver line	
import	240
reduction velocity.	280
reference level.	651
reflection amplitudes	
layer-show velocities	501
Reflexion	462
refraction data	462, 532
refraction seismics.	688
refraction tomography	669
Reinterpolate each 2D-line of a 3D-file	391
Remove header gain	344
remove non double picks	
picking	476
save picks	485
Remove range	395
Remove traces	395
remove zero traces	405
Replace traces.	398
report only construction data:	497
Resampling	339
residual statics	
CMP-stack	517
Resort, data	386
Resort-group	387
Resort-traceheader	386
reverse control	
traveltime analysis	690
Reverse polarity of traces	398
reverse shot.	697
Ricker	
FD-modelling.	661
Rotate 2 components	377
rotation	
3D-cube.	588, 605
Rresampling, data	384
Running average	408
S/R-distance, fileheader	291
Scaled windowgain(x)	347, 348

Scaling, import	250
Scaling, manual	268, 276
Scaling, plot	266, 276
Scaling, print	282, 283
Scan3D	560
Screen, delayed	208
Screen, split	277
scroll	
3D-cube	590
Scroll 3D-file	573
Scrolling	213
Scrolling, size	208
Scrolling, synchronize	208
second 3D-file	559
Second profile, show	277
SEG2	245
SEGY	245, 262
SEGY-Proceq	230
SEGY_PRISM	233
seismic crosshole data	
Original data	702
Seismic crosshole testing	702
semblance	430
Semblance analysis	533, 546
Sequence Processing	457
Sequence Processing, peculiarities	460
set pick code to range	
picking	477
Set to zero	399
several FileHeaders	324
several pages	
print preview	286
several single lines	
FD-modelling	662
shading	
3D-cube	591
Shaping	455
sharpen edges	
3d-datainterpretation	572, 575, 585
Shear waves - subtract traces	411
Shift-3DFile	391
Shot position, fileheader	291
Show markers	203
show picks	
interactive choice	203
show profile	
interactive choice	203
show rays	676
ShowTimeSliceInAddition	574
Signal-to-noise ratio, improving	339
single line	680, 686
FD-modelling	662
SIRT, Tomography	669

Slant stack, CMP-processing	518
Sorting, CMP-processing	508
Source couplings, compensation	347
Source-receiver-distance, correction	355
Space-increment, modelling	638
Spectral whitening	453
Spectrum, amplitude	375, 378
Spikes	333
Spiking-deconvolution	451
Split screen	277
Stack traces	411
Stacking, CMP-Processing	517
Starttime, move	355
Stat. correction, pick	361
Static correction	353
StaticCorrection/muting	351
Stolt migration	421
Subtract traces	411
Subtract-DC-shift	338
Subtract-mean (dewow)	338
Subtracting average	408
summary comment data	497
Suppress multiples	367
surgical muting	357
Swap 2 timeblocks	400
Swap bytes	245, 250
sweep	
crosscorrelation	339
swell correction	361
Swiss CH1903	259, 313
synthetic traveltimes	678
synthetic traveltimes	677
take over	
3D-picking	609
texture attribute	378
thickness determination	
picks	189, 468
Time cut	358
Time delay, correction	354, 355
Time-increment, modelling	638
Timedepth conversion	429
Timeslice calculation example	601
Timeslice calculation overview	603, 612
Timeslices	560
Timeslices, generate	598
timeterm analysis	690, 700
Tomography, modelling	637, 669
topographic correction	353
topography	
traveltimes	308, 319, 693
topography migration	422
Topography, modelling	659
total wavefield	

FD-modelling	662
Trace Interpolation/Resorting	380
trace normalize	347
Trace number	351
Trace spectrum	375, 378
tracegain	
plot options	277
TraceHeader axis	
view	220
Traceheader Edit	293
Traceincr-resampling	384
Traceinterpol-3DFile	391
Traces of a 2. line, display	214
Tracking	466
traveltime analysis 2D	688
traveltime branch	697
traveltimes	
assign	695
combine	697
put together	691
True amplitude information	344
Undo	647
model changes	658
picks	470
UpdateTraceHeaders	
Project fileheaders	325
uphole	462
use code	
3D-picking	608
use traceh.coord	
interactive choice	203
UTM-conversion	
traceheader	308, 327
Velocity adaptation	186-188, 462
modelling	643
Velocity model 2D, creating	549, 551, 552
Velocity model, adaption	542
Velocity model, CMP-data	532
Velocity model, CMP-processing	508
VelocityAdaptationPanel, display	214
View MenuItem	214
Vrms velocity	536
VSP (Vertical Seismic Profiling)	
CMP velocity analysis	532
VSP (vertical seismics profiling) measurement	462
VSP corridor stack	
1D-filter	390
Walsh bandpass	340
wavefront inversion	677
wavefront-inversion	688
traeltime analysis	699
wide-angle	532
Wiggleattributes	275

Wigglemode	266
WiggleWindow of actual trace, display	214
Windowing 3D-file	583
x-distance decay(db)	
gain	348
x-project	642
XFlip-3DFile	391
XFlipProfile	386
xy-projection	
picks	203
XYScaledPlot.	266
XY€exchange	387
YFlipProfile	386
Yo-Yo section	388
z-project.	642
Zero level.	271
Zero mean	338, 634
Zero traces	404