GLOBE Claritas

seismic processing software

GNS SCIENCE
TE PŪ AO

2D LAND TUTORIAL

VERSION 6.8

Connect. Build. Grow.

TABLE OF CONTENTS

1 Getting Started

This tutorial is designed to get you up and running quickly with the GLOBE Claritas^{TM} seismic data processing package. It's based around a 2D land seismic line from onshore Taranaki, New Zealand.

While it is straight forward enough for inexperienced processors to get good results, there are also enough challenges to make it interesting for more experienced people.

The tutorial is not a comprehensive showcase of all the functionality and modules in the software or a complete guide to 2D land processing, however it does explore some of the complexities of both.

As well as this document, you will need:

- To have installed GLOBE Claritas V6.8 or higher
- To have the Land Tutorial dataset V68_2DLAND.ca, in a suitable folder on your system

If you are missing either of these, please contact **claritas.support@gns.cri.nz**

2 The Launcher, Getting Started with Projects and Finding Help

2.1 GLOBE Claritas Basics

GLOBE Claritas is a file based seismic processing system – there's no database.

There are broadly three types of files that we use; seismic data, "meta data" and processing flows.

Seismic data is usually stored in an open format called HDF5; this is a hierarchical file that you can think of as a hybrid between a file and a database – so for example we can access and modify trace headers without rewriting the file in some cases.

Meta-data is things like velocities, mutes, processing "gates" and so on; these are mostly stored as ASCII format files, although geometry is held in a binary structure. Where these files can vary in time, space or with trace headers they are in a spreadsheet format, and there's a spreadsheet editor to help you create, build and modify them.

Processing flows are also stored in an ASCII format, however there is a form-based interactive job flow builder that lets you create or modify these files.

All the interactive analysis, display and data management tools work with these three file types.

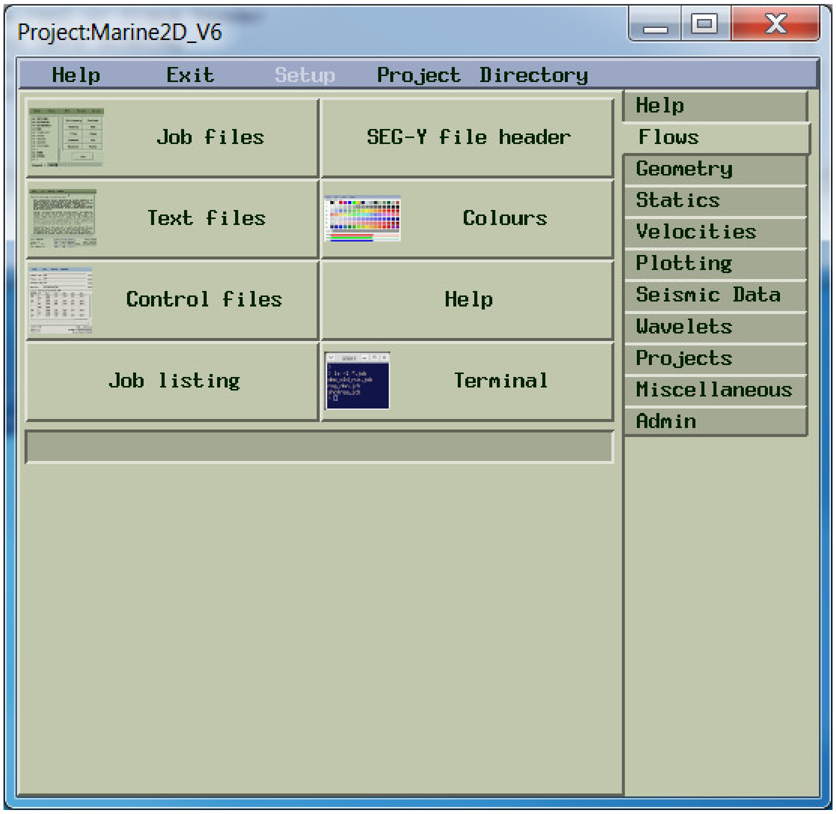
2.2 The Launcher

GLOBE Claritas^{TM} is a suite of applications and utilities for analysing and processing seismic data. These tools all read and write a common set of formats, which are based on "open standards" like SEGY, HDF5 and ASCII. There is no underpinning database, so other applications can read these files directly.

You can get access to all the applications and utilities via the Launcher.

On the Windows operating system you can start the Launcher by **clicking on the GLOBE Claritas™ icon** on the desktop; on Linux simply type '**launcher'** at the prompt in a terminal window.

The Launcher tabs (down the right side) allow you to select the different types of application or utility. In some cases the same application may be under several tabs but operating in different modes.



The GLOBE Claritas™ Launcher

Q. Do I have to use the Launcher?

A. No, you don't. You can also start applications and utilities from the command line or prompt if you want to. The launcher does help you to manage your data by grouping it into "Projects", so it is recommended.

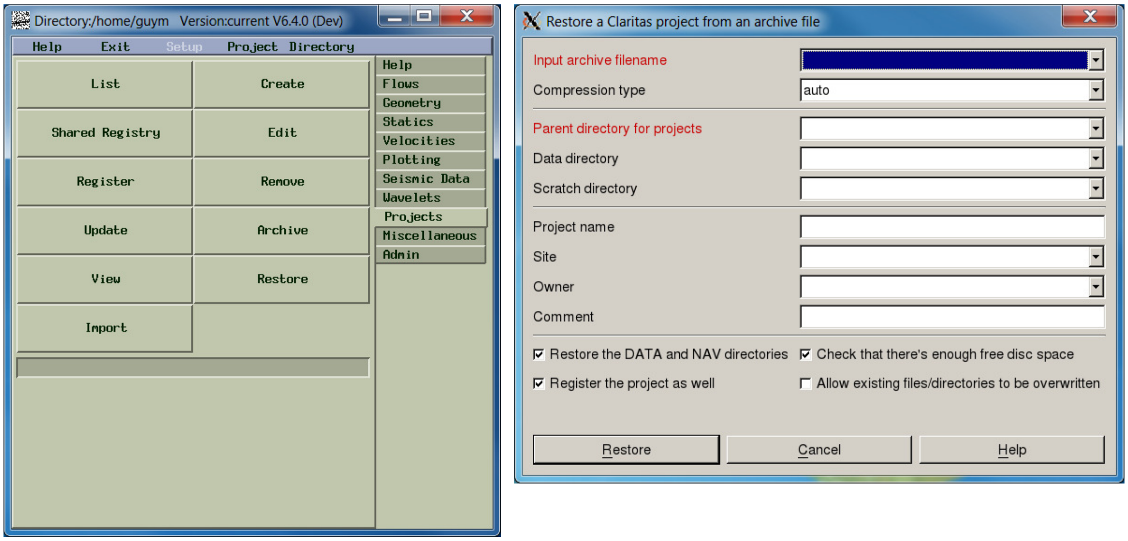
2.3 GLOBE Claritas Projects and the Land Tutorial

To help you to keep your processing work organised, GLOBE Claritas™ groups files into "projects." Each project is a fixed structure of folders or directories, with a defined home for each type of file.

You don't have to work with projects, however when you do work in a project, the folder names are all managed for you automatically – each tool knows where to look for a given type of file, and where to write it out to.

The land tutorial is an **archived project**; this can be restored by a special utility under the "Projects Tab"

- Click on the Projects tab
- Click on the Restore button



The GLOBE Claritas launcher showing the "Projects" tab (left) and the "Restore Project" dialogue (right)
In GLOBE Claritas forms, only items in red are mandatory, the others can be left blank or default.

The only things that have to be specified on the "Restore Project" dialogue box is the name of the parent directory (or folder) for the project, and the name of the project archive being recovered. The "Browse filesystem" option, select the down arrow on the right, allows you to search the file system.

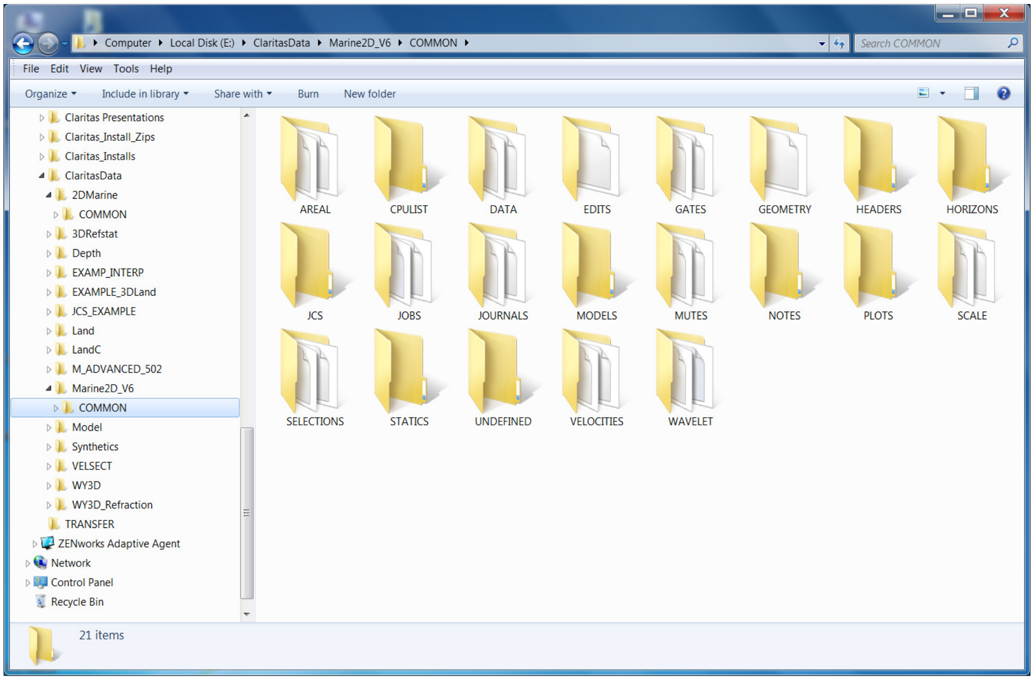
The project archive should be called **V68_2DLAND.ca**, if you don't have access to this please contact your system administrator, or the support team via claritas.support@gns.cri.nz.

- Click on Restore on the Restore Project Dialogue

The utility will now unpack the project archive and store it in the directory you specified. There is a list of projects called the "project registry" – the new project will be added to this. If you didn't specify a name and so on for the project, it will be read from the archive as well.

- When the project has restored, close the dialogue

Here's an example of a project structure, on a Windows system:



The directory structure for a GLOBE Claritas™ project (Windows 7); in this example the project name is MARINE2D_V6, below this is the COMMON directory and then directories for each of the main data and file types. This is created and populated automatically by the project restore process

You shouldn't usually need to go into the project structure, but it can be useful to know where individual files are placed.

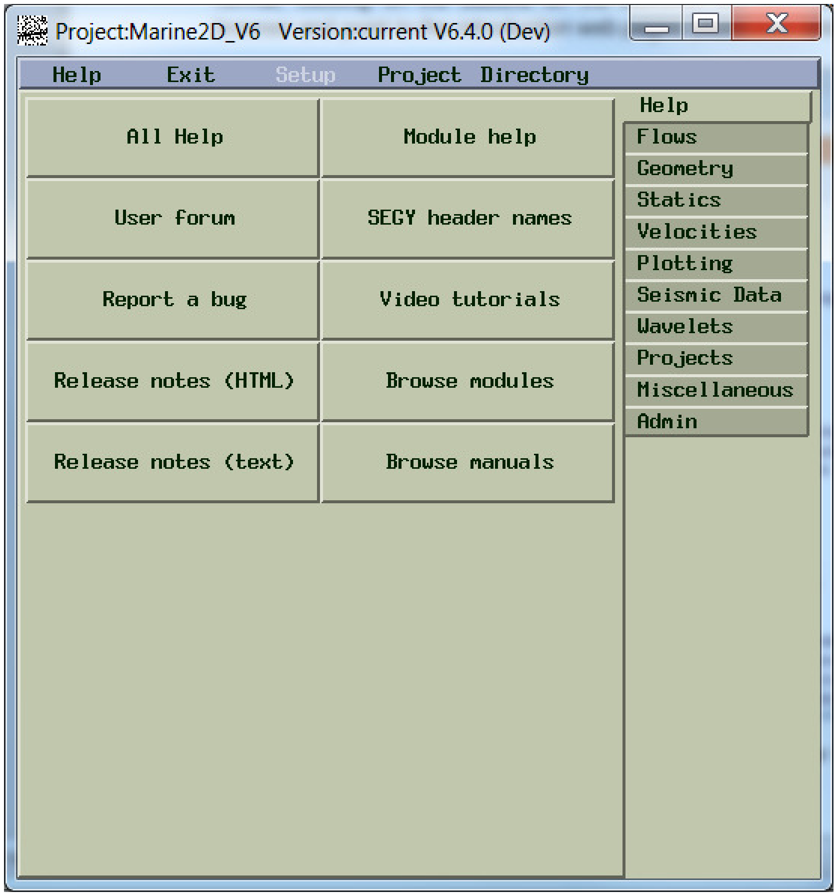
Now you can select the project you restored to be active

- **Click on the Project option in the toolbar menu at the top of the Launcher (between Setup and Directory). In the new window that opens select the project you just recovered. The 'Filter' button can be used to search the list of projects**

When you do this the name of the project should appear on the Launcher.

2.4 Getting Help

The Launcher also has a tab marked "Help"



The Help tab on the Launcher, allowing access to text based or web-based manuals and support information.

The "Browse modules" and "Browse manuals" buttons open up the web-based version of the manual and support information, which has full search and cross-linking of key topics.

The same information is also available in the text-based application "Seishelp" through the "All Help" button. This is also fully searchable, and uses a menu-driven approach.

This help also functions in applications and utilities, providing a third way to access the same information.

2.5 Comment on Naming Conventions

While the project system takes care of a folder structure, seismic processing generates a lot of files.

For a testing sequence like this, one naming convention we have found useful is to simply number each processing flow, starting from 01. Where a flow has an output file, or uses a meta-data file, this is also given the same number.

In this way you preserve a strong link between the processing flows, the files they use, and the files they create – and one that is listed out logically when you review a directory.

3 Seismic Line PC96-08 – Viewing the Raw Data

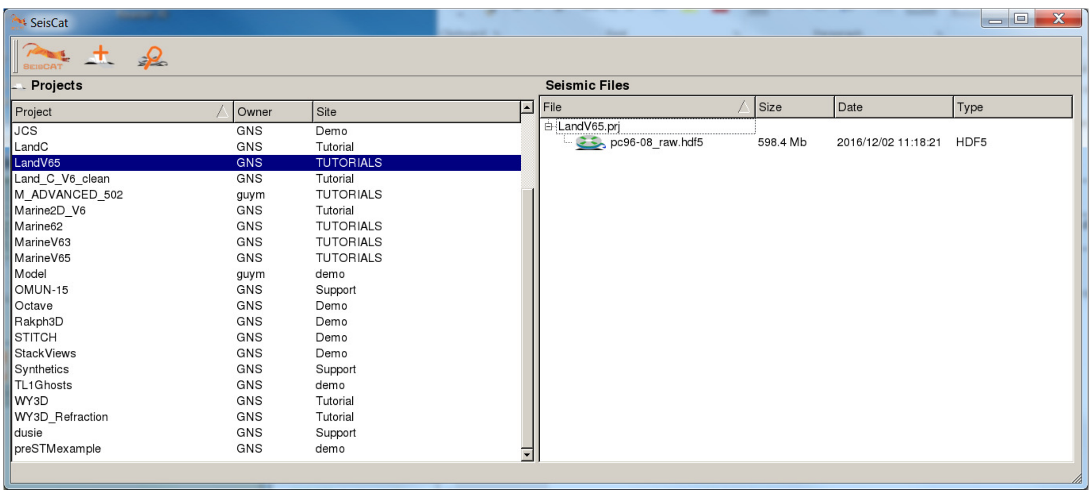
3.1 SeisCat and SV

Once you have unpacked the project, you can start to look at the raw seismic data. The data has already been converted into the GLOBE Claritas **HDF5** format.

- **Click on the Seismic Data tab on the launcher**
- **Click on the SeisCat button**
- **Update the view by pressing this icon :** 

SeisCat is a data management tool that automatically locates seismic files in your projects – in SEGY, SEGD, GLOBE Claritas SEGY or HDF5 format, and allows you to sort, display and interact with them.

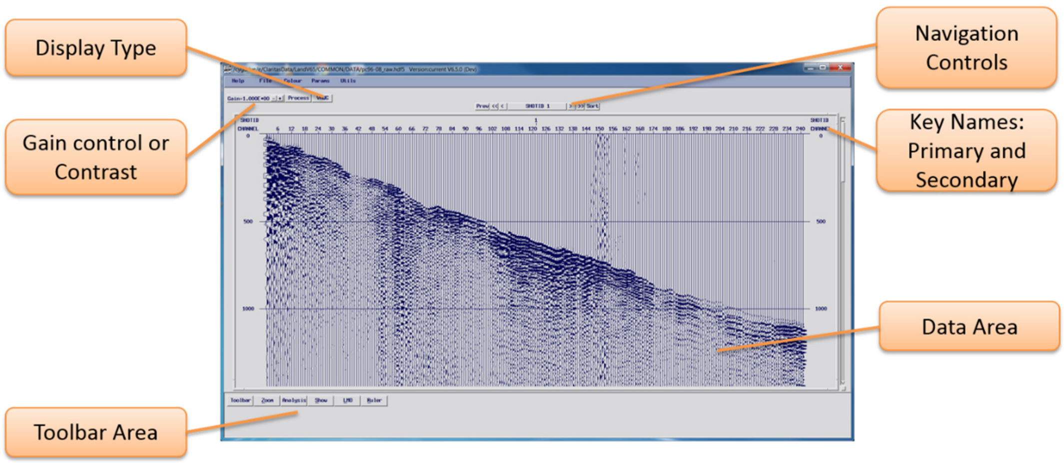
The initial view will look something like this, if you already have some projects defined (however, this tutorial might be the only project you have at this stage):



The SeisCat display. Projects are on the left-hand side, with the data files displayed on the right

- **Highlight the file pc96-08_raw.hdf5 by clicking with the left mouse button**
- **Right click to get the interaction menu**
- **Select "Open with SV"**
- **When the dialogue window opens, enter "OP1" into the Output Files and ignore the other parameters for now**
- **Click Ok, and the press "Continue" on the pop-up window about COORD_SCALE**

SV is one of the seismic viewing tools in GLOBE Claritas^{TM}, it is designed mainly for creating some key types of information used in processing, such as horizons, picking "first breaks", creating mute files and so on, which is why it needs an output file name.



The SV seismic display tool displaying the dataset PC96-08_raw.hdf5

A seismic trace has a number of "trace headers" that are used to help identify it. There are some standard headers – ShotID, Channel and so on – which are commonly used, but in GLOBE Claritas^{TM} the user can also make new headers.

Data are usually displayed in groups, defined by a trace header. Within a group (also called ensembles or gathers) data can be also ordered by a sub-group.

The groups and sub-groups are referred to as the "Primary key" and "Secondary key" of the dataset.

In this dataset the original order is to have a Primary Key of SHOTID, and a Secondary Key of CHANNEL. SHOTID represents the original file number on tape, and the CHANNEL is the recording channel number.

You can find a full list of the seismic trace headers that can be used for Primary and Secondary keys in Appendix A of the Manual.

3.1.1 Exploring the Data – PC96-08

Line PC96-08 was shot in 1996 for Petrocorp Exploration Ltd., over an overthrust in a producing oilfield in the Taranaki region of New Zealand. This data is available from the New Zealand Ministry of Economic Development under the New Zealand Open File system.

The Taranaki region has significant relief, complex velocity structure, and is prone to significant static shifts arising from strong near surface depth and velocity variations. Like many Taranaki datasets, this seismic line has a low signal to noise ratio, and limited bandwidth, and so presents some interesting processing challenges.

The Main acquisition parameters are:

Source Type:	Dynamite
Shotpoint Interval:	40 metres nominal
Receiver Type:	Geophones

Group Interval:	10 metres
Max Number of Groups:	480
Original Data Format:	SEGD 3480 Cartridges
Sample Interval:	4 milliseconds
Record Length:	6132 milliseconds

The line is 8.6km long, comprises 221 shots, with a maximum shot fold of 480 channels, and a resulting maximum CDP fold of 130.

- **In the open SV window click on the Display Type button to go from VAWG[1] display to VD[2]**
- **Use the – and + keys on the keyboard to change the display scales**
- **Use the < and > keys on the keyboard to change the horizontal display scales**
- **Click on the colour bar (in VD mode) to change the colour scheme (LMB for a menu)**
- **Use the navigation controls (<<, <, >, and >> buttons) to step through the data in either direction**
- **Click on 'SHOTID … ' to enter a shot number to skip to.**

Geophysics Comments:

- As you step through the shots note how the "apex" of the shots varies. Initially the shots are fired into the spread from one end, however we move towards a "split spread" with the shot in the centre, and eventually the shots "run-out" at the end of the line.
- Direct and refracted arrivals are only roughly linear; this is a result of a combination of the rugged topography the line is shot over and the variations in near-surface velocity
- The channels closest to the source tend to be swamped by "ground roll" – that is energy propagating in the near surface. In-fact parabolic reflections are hard to spot!
- The apex of each shot is not quite at time zero, because the explosives are at depth, and the receivers on the surface, creating a small delay.

As you move the cursor around the plot, the trace amplitude, secondary key and the time in milliseconds are displayed in the top left corner.

You can apply some simple processing to the data while you are viewing it. At the moment the data is completely "raw" so the amplitudes are highly variable. We can balance the amplitudes by applying an AGC (automatic gain control) to the data.

- **Click on the Process button**
- **Type AGC into the Processing sequence parameter**
- **Press OK, and then Ok again to accept all the defaults**

[1] Variable Area Wiggle – amplitudes are represented by the size of the wiggle, with positive value shaded black
[2] Variable Density- amplitudes are mapped onto a colour palette

Geophysics Comments:

- Automatic Gain Control (AGC) is used to control amplitudes in a processing flow
- Data is scaled within a window so that the RMS value of the data is 1; typically this means the majority of the traces lie between amplitudes of +/- 5
- The window slides down one sample at a time, with a different scalar calculated for each window.
- AGC is a crude tool, it is good for showing up strong signals and weak ones at the same time, but relative amplitudes are lost.
- Without an AGC, we might focus only on the strongest events; with an AGC applied, we may miss key amplitude anomalies or artefacts.

There are many more things that you can do in SV, including picking mutes and first breaks which we'll look at later.

4 Creating and Running Processing Flows

4.1 Why Use Processing Flows?

While it is useful to be able to apply a single processing stage like AGC interactively, more complicated processes can take time. It's usually much more efficient to combine a series of processing steps into a "processing flow" that can be run in the background while we do other things.

There's essentially two type of processing flow in GLOBE Claritas; those that run through to completion, and those that create an interactive display on the screen.

In general, the processing flows that run to completion produce outputs that we can then look at or analyse with an interactive tool later on. The main reason to use an interactive display is that it allows a quick and easy way to test different processing sequences.

Where these tests take a long time, however, it can make more sense to run them as separate processing flows with no interactive output.

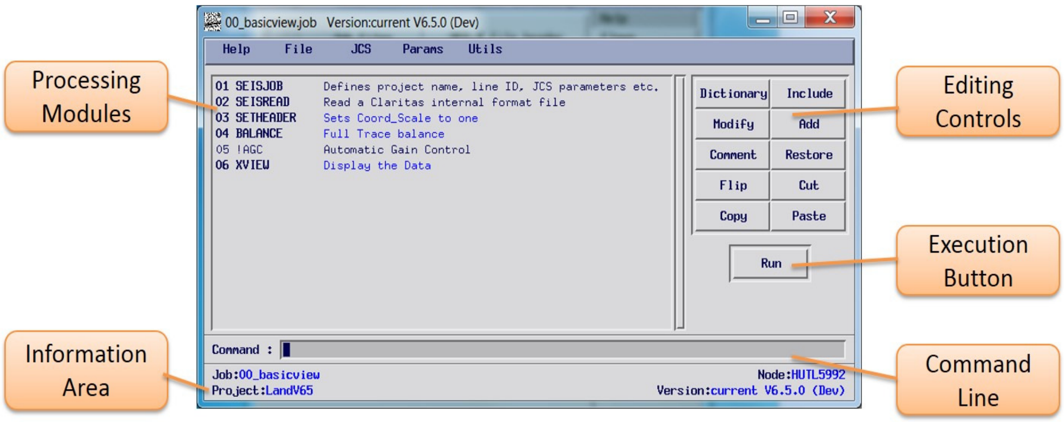
4.2 Getting Started – the XSJE Seismic Job Editor

A processing flow in GLOBE Claritas is also called a "job". Each job is effectively a short program, written in a specialised high-level language designed for seismic processing.

While they are saved as simple ASCII text files (and you could manually edit them if you really had to) there is a special editor called XSJE that makes this process much easier.

- **On the Launcher, select the Flows tab**
- **The button marked "Job files" starts the job flow editor**
- **Select the file called 00_basicview.job using the "Job file" button to bring up a file browser**

Note that XSJE automatically "looks" for processing flows in the JOBS folder within the project structure. Any changes will be saved in that same folder.



The XSJE Seismic Job Editor. In this flow there are six processing steps or modules, with the 5th step, AGC, currently deactivated.

Each processing flow comprises a series of steps, called "modules" which act on the seismic data in a different way. In this flow there are 6 steps in total, numbered from 01 to 06.

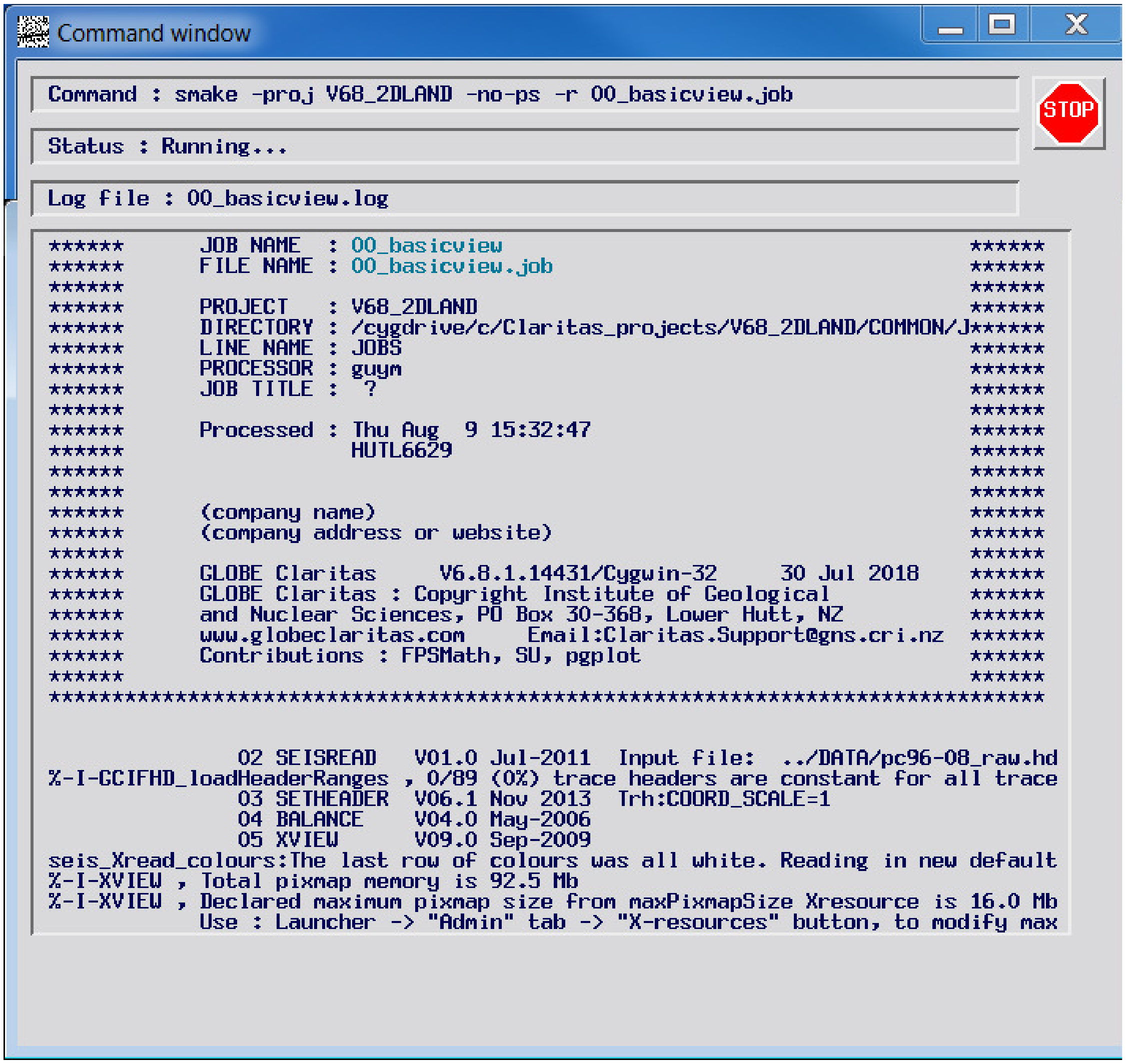
Each module has a unique name in capitals, followed by a brief comment that describes what the module does. These comments can be edited just by double clicking on them so that you can make your own notes on the flow. Comments in blue are ones that have been edited.

The first module "SEISJOB" is always present, and is used to store key "metadata" about the processing flow.

The information area helps you to keep track of what this flow is called, what computer it will run on, as well as the current project – all of which can be useful if you are working on multiple projects.

- **Click the Run button**

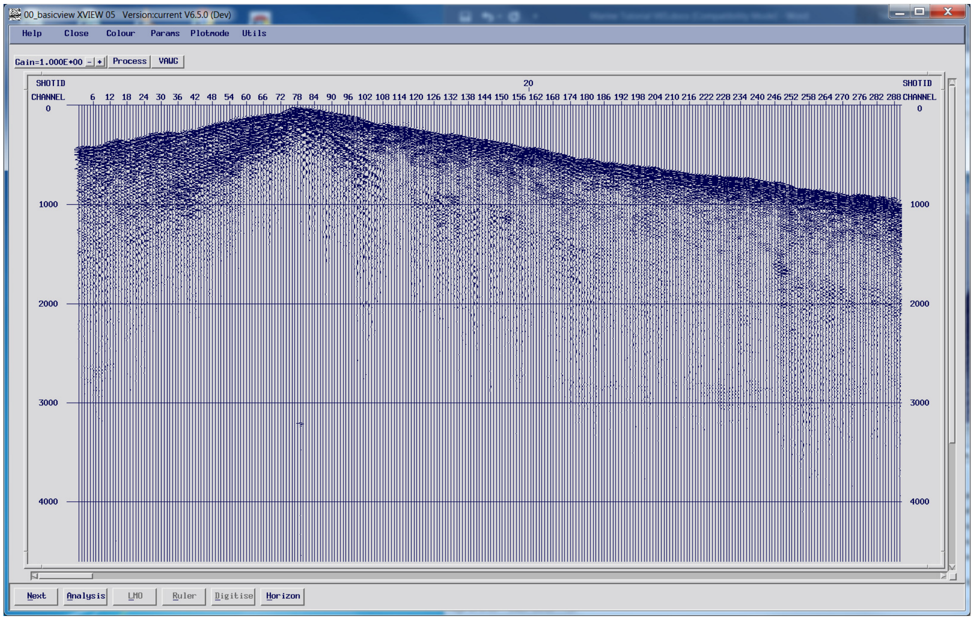
When you run a processing flow, it is checked, complied into an application and then run on the computer. A pop-up window called the "Command Window" appears which gives you information on this process.



The Command Window which appears when a job is running, letting you know how things are progressing. Warnings appear in purple, with any error messages in red. Support and data files are highlighted in blue and act like hyperlinks when job flow not active, Right Mouse Button (RMB) brings up a menu where the user can select appropriate tools to view/edit file.

This processing flow includes an interactive display, which is the module called XVIEW at the end of the processing sequence.

An XVIEW display will also open; it is similar to the SV display, only without any toolbar options or navigation controls.



The XVIEW display generated by the processing flow 00_basicview.job

When you run the job, the XVIEW window will appear within a few seconds and a grey vertical progress bar (on the left-hand side) will indicate how much of the data has loaded. After enough data has been collated to fill the display, a green bar will show the progress of rendering the display. On most systems this will happen almost too fast to see, but if you have a lot of modules in the flow or a slow display speed from a remote location, you may see both the grey and green coloured bars.

The data will start to display, but you will need to use the scroll bars to see it all. Note that once the data has all been loaded for display, the cursor changes shape to a cross.

As you move the cursor around the plot, the trace amplitude, secondary key and the time in milliseconds are displayed in the top left corner. This processing flow is configured to only show a selection of the data, with ten shots in total in the display.

You can change this display in the same way you modified the SV display earlier, however you cannot navigate, or change the data selection in an interactive flow.

- **Click the Next button**

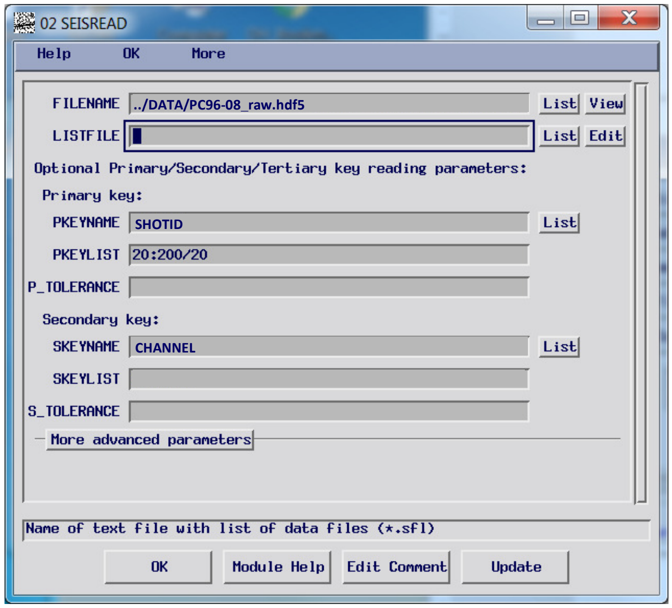
The processing flow will complete when you do this. The command window will remain until dismissed, allowing you to see some performance metrics such as how many traces were passed through each module. These can be helpful in troubleshooting flows that don't work as anticipated.

4.3 How to Edit and Build Flows

Dismiss the command window, and go back to the processing flow display.

- **Click the module 02 SEISREAD**
- **Press the Modify button**

Each module has a series of user-defined parameters that control how it behaves. The SEISREAD parameters control the order in which the processing flow will read the data, and can be used to make selections.



The SEISREAD parameter form

In this example you can see the filename that is being read in, which is the same file that we looked at earlier. The PKEYNAME and PKEYLIST parameters define the primary key to be read, as well as which primary keys we are interested in.

If you click on the 'Module Help' button you can see a description of how the PKEYLIST parameter works; in this case we are telling the module we want to read from SHOTID 20 to 200, with an increment of 20.

We also define the secondary key (CHANNEL).

Click on OK to close this form. Note that you can also open a parameter form directly just by double clicking on the module name, which is usually more convenient.

- **Click the module 04 BALANCE, and holding down the left mouse button also highlight the module marked 05 !AGC**
- **Click on the Flip button**

Modules that are "deactivated" are shown with less emphasis (not bold) and a "!" in front of them; we have just deactivated the BALANCE module and activate AGC. Run the job again to see what effect this has.

If you highlight the two modules and click the right mouse button, you get access to a menu which includes the FLIP option.

This is useful in trouble-shooting flows; you can deactivate a module without losing the parameters you have entered.

4.4 Adding and Removing Modules from a Flow

The XSJE flow editor allows you to remove one (or more) modules with the "Cut" button – just click on the module name (or hold down the left mouse button to highlight a group of modules) and then click on Cut.

You can also "paste" this selection – complete with parameters – elsewhere into the flow.

The Add button is used to add a module into the flow.

- **Click the module 05 AGC**
- **Click the Cut button to remove this module**
- **Click on the Add button**
- **Click on "Alphabetic Listing" and click OK**
- **Select AGC from the list, and click OK; the pointer will change shape (to a pencil icon)**
- **Position the pointer between the module 04 (BALANCE) and 05 (XVIEW) and click**
- **The module AGC will be inserted in position 05, and you can edit as before**

Expert User Tips:

- You can double-click to avoid having to click 'OK' after each step above.
- You can also insert a module using the keyboard by typing commands.
- In the 'Command' field, type "add AGC 05" and press enter to add the AGC module between the BALANCE and XVIEW modules.
- Use **append** or **app** to add a module to the end of the flow; eg **app AGC**.
- If you just type **app** or **add** this opens a selection dialogue, exactly as if you had pressed the buttons.

You'll notice how modules could also be listed out by type as well as alphabetic listing; there's over 200 different processing modules you can use, so sometimes looking via "category" can be useful.

Expert User Tips:

- To look at the files created while we have been working, open a terminal window from the Launcher by clicking on the 'Terminal' button on the 'Flows' tab.
- Here you can enter UNIX/Linux commands – even if you are running GLOBE Claritas™ under Windows. If you type "ls -latr 00_basicview.*", you will see a complete list of all of the files associated with running this job. The ***.job** file contains the workflow, and the ***.log** file is the log output from the **Command window**.
- Note that each time you run a job, the log file will have the same name, even if you have added modules or changed parameters. The current log file is called 00_basicview.log; if the job is re-run a new log is created and the old version is renamed to 00_basicview.log~. When running a testing sequence it is a good idea to save each job under a different name as you edit or change the flow, using the "File: Save As" option. This avoids losing old tests and examples.

5 Survey Geometry – Positioning the Seismic Data

The only positioning information we know about our seismic data at this point is the SHOTID and CHANNEL numbers.

We need to know how these numbers relate to surface locations – including the elevation – so that we can form an image of the sub-surface.

While modern "node-based" recording systems might have the Easting, Northing and elevation for every receiver and shot location automatically recorded into the seismic trace headers, it's more likely that we will need to start from separate "survey files."

The SEG standard for these files is called SPS, and was originally developed by Shell. In SPS format, each line is described by three ASCII files:

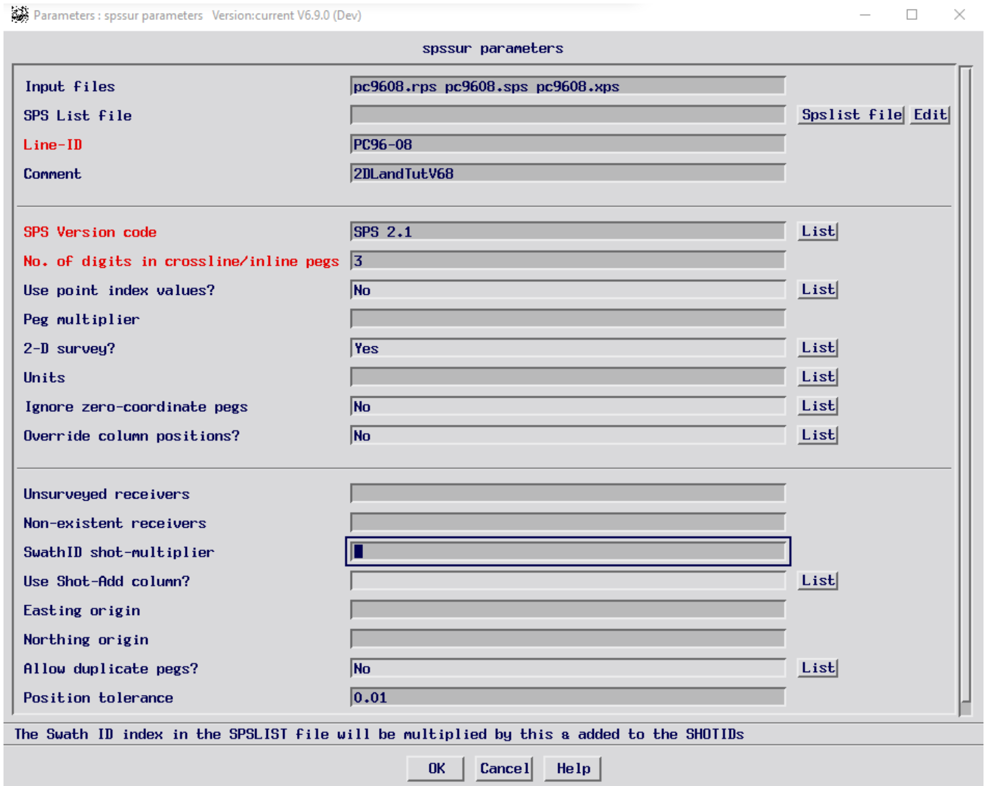
- the "shots" file with a .s extension, which gives the X,Y,Z position of each shot location
- the "receivers" file with a .r extension, which gives the X,Y,Z location of each receiver location
- the "relations" file with a .x extension, which maps the file (SHOTID) and channel numbers to the shot and receiver locations

In GLOBE Claritas, we create a geometry database containing all the positioning information for the seismic traces, and then merge this data into the seismic trace headers.

5.1 Building a Geometry Database from SPS Data

The GLOBE Claritas Geometry application is used to build and visualise geometry databases.

- **On the Launcher, select the Geometry Tab**
- **Click on the "Set-up from SPS" button**
- **Complete the form as below:**

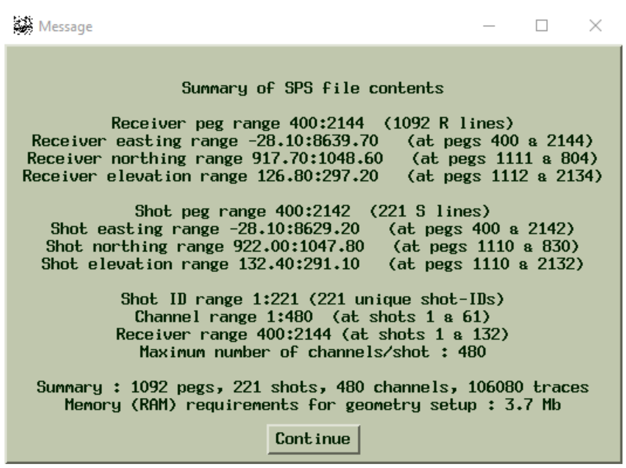


Setting up the survey information from SPS files

Like most of the SEG standards, variations exist to cope with situations in the field that were not anticipated at the time when the standard was formed; while there are options to address many of these issues, they are not required in this case.

- Click on OK

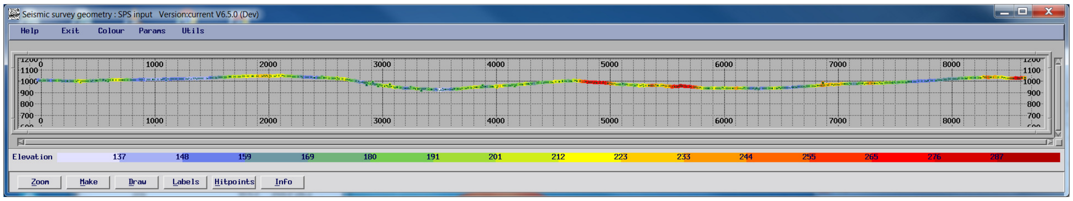
The application will scan through the SPS data, and give a short report on what has been located as well as any anomalies identified in the file. In this case, the Easting and Northing values are relative, not absolute.



Summary produced on loading the SPS data into the geometry application.

- Click on Continue, and then in the next pop-up message again click on Continue

The next thing you should see is the survey information displayed in map form. GLOBE Claritas calls each unique shot or receiver location a “survey peg”, and the initial display is of all the shot and receiver peg locations colour coded by their elevation.



Initial map display of the survey data, with the survey pegs colour coded by their elevation.

5.2 Visualising the Survey Data

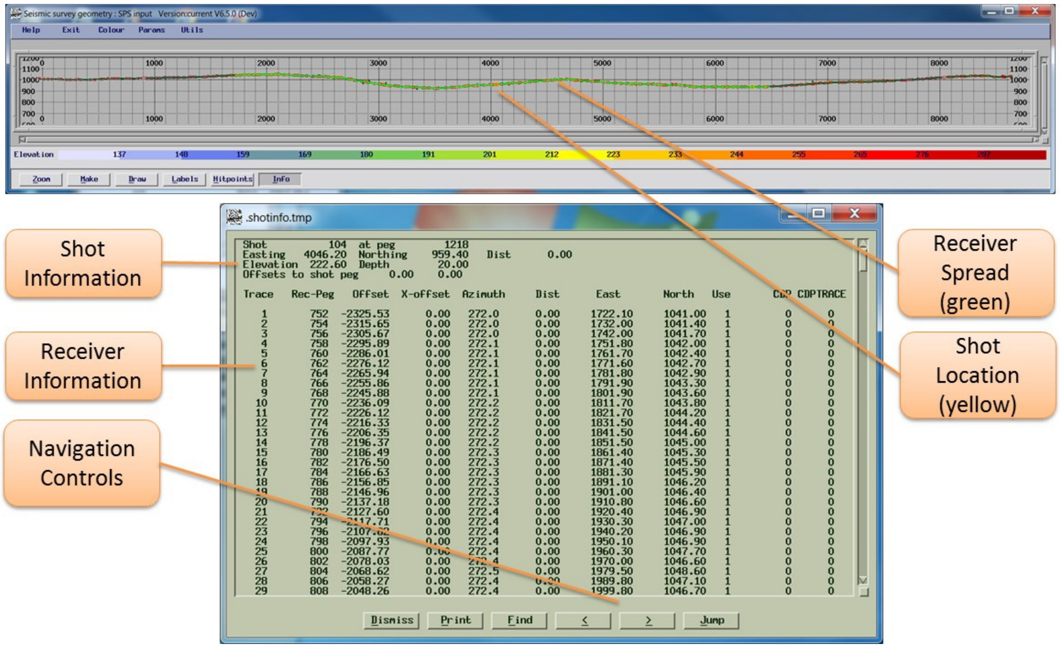
The “Draw” button gives you access to various ways to visualise the survey information, with the “Labels” button providing annotation.

The 3D perspective view is particularly useful to understand the complexities associated with the survey, especially where the seismic line has followed roads or paths over rough terrain.

You can use the “Info” button to explore the data in more detail.

- Peg Information gives basic data about the X,Y,Z locations of the shot and receiver pegs
- Shot Information gives the shot and receiver locations for a given file, as well as the offset and azimuth of each trace. The Shot selected is indicated on the map (yellow square), with the receivers (if displayed, green crosses) also indicated. There are navigation controls on the shot information window that can be used to step through shots.

You can use the same keyboard controls to change the plot scales as you did in XVIEW/SV (+, -, <, >), or adjust them under the Params menu.

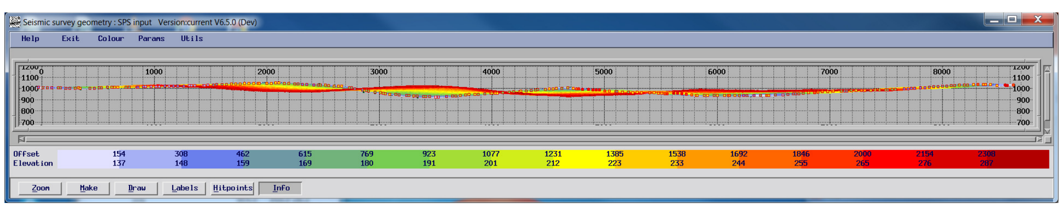


Using the 'Shot information' option under the 'Info' button to examine the survey information.

5.3 Creating CDPs – Crooked Line Binning

As part of processing the seismic data we need to group the traces into a series of "common depth point" (CDP) gathers. We do this by creating a series of CDP "bins" along the line, and collecting traces into a CDP bin if the midpoint between the shot location and receiver location falls inside the bin limits.

You can display the trace midpoints using the "Draw Midpoints" option; we need to make sure that the CDP bins sit on a line that runs through the centre of the "cloud" of midpoint data.

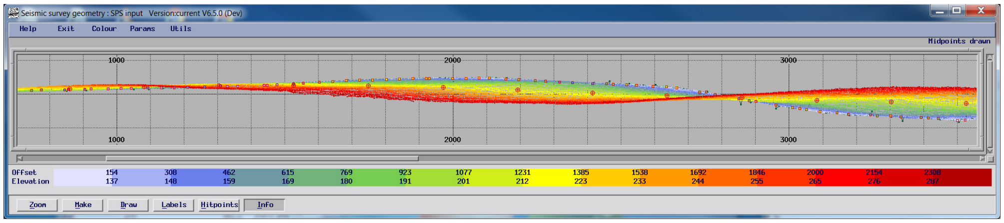


The source-receiver midpoints displayed, colour coded by the distance between the source and receiver (offset); the CDP bins need to run through the centre of the "midpoint cloud"

We define a path for the CDP bins to follow by creating a number of "**hitpoints**"; the CDP bins will then follow the best fit spline curve through those hitpoints.

GLOBE Claritas can create hitpoints automatically, or you can pick them manually.

- **Click on the Hitpoints button**
- **Select Make automatic picks**
- **Use 10 hitpoints**



Detailed view of the midpoint cloud and the automatically generated hitpoints. The hitpoints are red crosshairs

Now the hitpoints are in place, we can make the CDP bins.

- **Click on the Make button**
- **Select Wiggly-Line CDP gather**

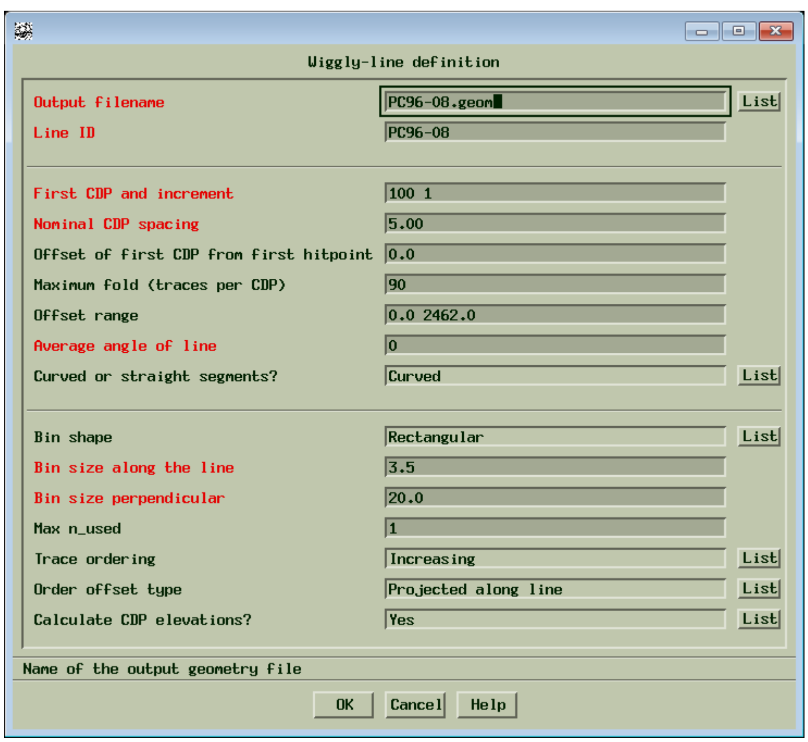
For the wiggly line gather we need to set a few things.

Firstly, we need to define the CDP spacing; this will be calculated as half of the mean receiver spacing but it might not be exact.

Secondly we need to set a maximum fold for the bins.

Finally, we need to set the bin shape; the bins might overlap if we turn a sharp corner (controlled by "bin size along line") and we need to give the bins a width.

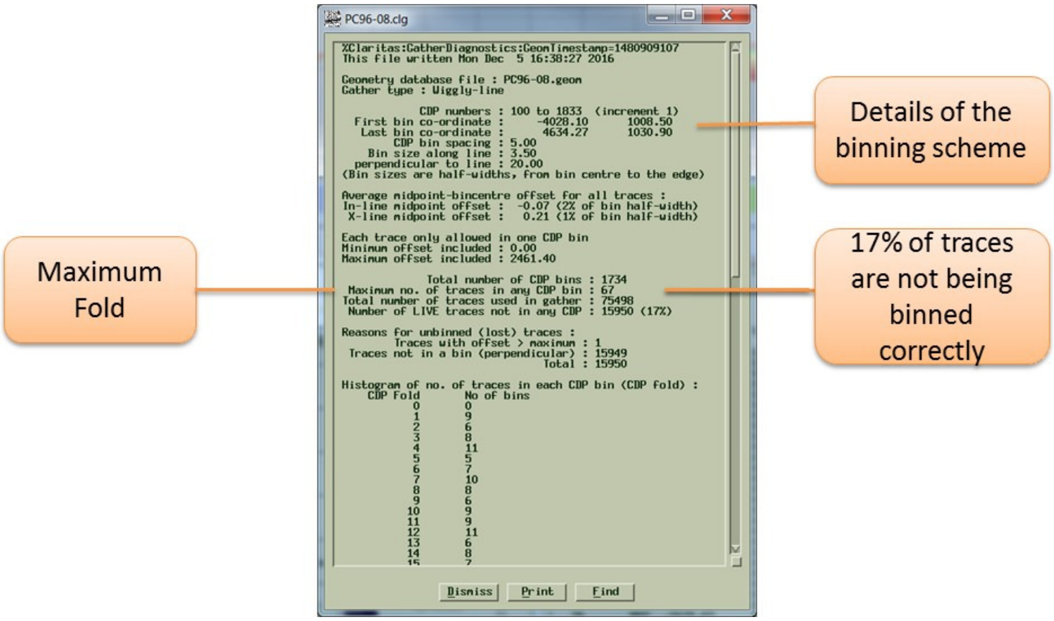
For now, use the following values:



Initial crooked line binning parameters.

- **Press OK**

You should see a warning message, followed by some binning statistics.

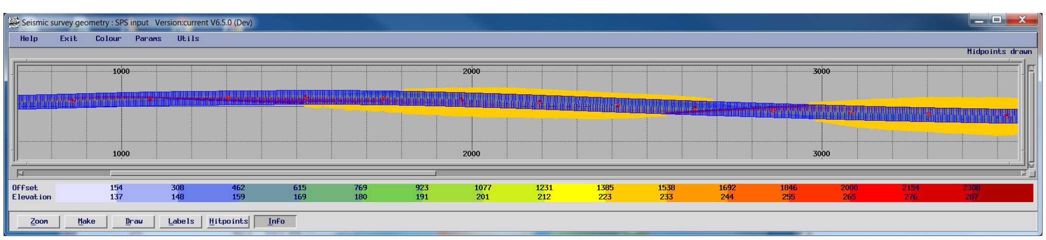


Binning statistics from the default parameters; the statistical breakdown highlights a problem with the binning – 17% of the traces are falling outside of the bins. The number of traces in each bin is termed the "fold" of the CDP.

The CDP binning parameters we've chose haven't worked very well; we're losing 17% of the traces.

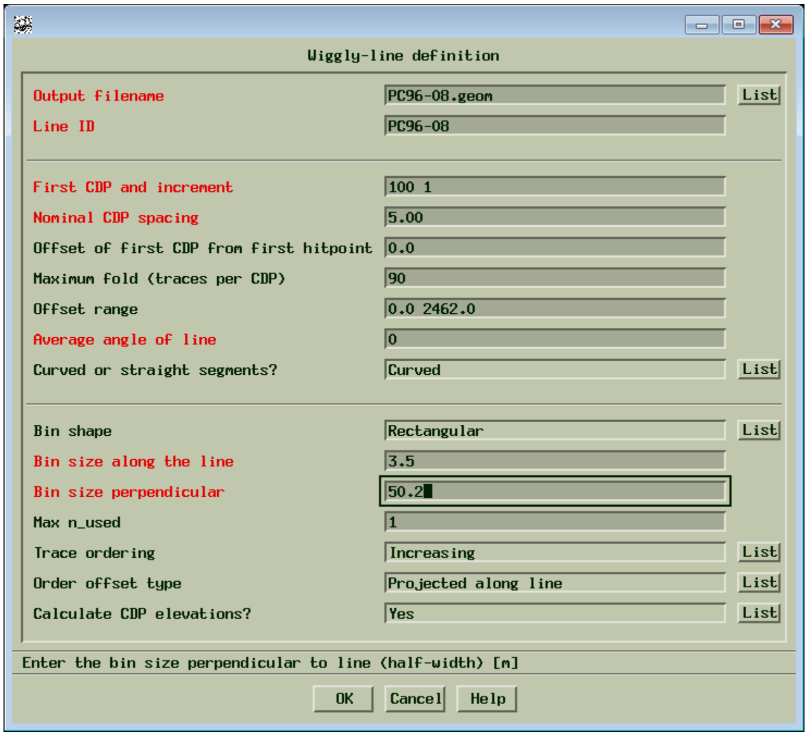
- **Click on the Dismiss**
- **On the main map, click on Draw**
- **Turn on 'CDP bins', 'Trace midpoints' and 'Lost traces'**

This kind of display is useful to see how and why we are losing traces from the binning scheme. The missing traces are colour coded yellow – in this case it's obvious that our CDP bins are not large enough. We need to more than double the bin size perpendicular to the line.



Detailed view of the midpoint cloud, the CDP bins (blue rectangles) and the lost traces (yellow) which are not falling into any bin. We need to use larger bins.

Go back and set up the wiggly line CDP binning scheme again (Make – Wiggly-line CDP gather) but use the parameters below:



Revised crooked line binning parameters; the 'Bin size perpendicular' has been extended to 50m

There are now more options available under "Draw" – you can look at the areal fold distribution, which colour codes the CDP bins based on the number of traces in each bin, as well as the fold histogram, which shows the traces in each bin colour coded by offset.

The CDP information is automatically saved in the geometry database we created when we loaded the SPS data.

5.4 Managing Elevation Changes Using Floating Datum

As we have seen from the elevation visualisation, the seismic line crosses quite rugged topography; we've also seen the effect this has on the seismic data earlier.

To address this we need to use a "floating datum" – essentially a smoothed version of the elevation profile – which will replace the actual topographic surface during processing. We can correct the data to be relative to this datum using a series of static shifts, defined by the surface location.

Geophysics Comments:

- We need to correct the data for the variation in elevation of the shots and receivers
- Normally this means using a new, fixed datum plane, and correcting the seismic data as if the shots and receivers were on this plane
- The corrections are time shifts based on a fixed velocity, called the replacement velocity. This is a good approximation if these "static" shifts are small (2-3 samples)

Geophysics Comments:

- Where we have rugged topography, like on this line, the shifts can be large. To address this instead of a fixed datum plane, we can use a "floating datum" during processing
- The floating datum is a smoothed version of the topography. At the end of the processing we can then convert from the floating datum to a fixed one.

- **Click on 'Make'**
- **Select 'Floating datum statics'**
- **Set the 'Fixed datum' to be 300m and the 'Replacement velocity' to be 2300 m/s**
- **Use a large 'Smoothing filter length', eg 149 points**

The fixed datum is usually set up to be above the highest elevation on the survey, as this will be "time zero" on the final seismic sections.

The floating datum is saved as static shifts tied to the CDP number of the traces.

You'll get a quality control display of the floating datum where you can check to ensure it is smooth; you can dismiss this, then exit from the Geometry tool.

6 Merging the Seismic Data with the Geometry

6.1 Running a Flow to Add the Geometry

In GLOBE Claritas, the seismic data is stored as files. This means we need to create a version of the raw input data with the geometry information merged into the trace headers.

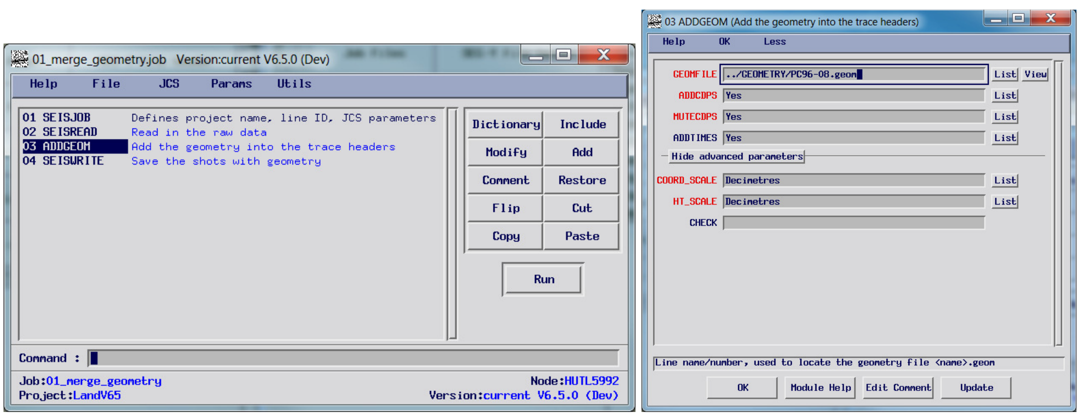
Where in the original data we just had the file number (SHOTID) and trace within that file (CHANNEL) defined, after merging the geometry we will have configured the following trace headers:

Seismic Trace Header Name	Usage
COORD_SCALE	Trace headers by default are integers; the coordinate scalar is used to allow for greater precision. The default is to use decimetres (or deci-feet) which corresponds to a coordinate scalar of -10.
COORD_UNIT	Defined when we created the geometry database.
HT_SCALE	This defines how elevations or depths are stored, in the same way that the coordinate scalar is used. Default is -10 for decimetres.
OFFSET	The distance between source and receiver, stored as an integer scaled by the coordinate scalar.

CDP, CDPTRACE	The common depth point (CDP) number and trace-count within the CDP assigned to the trace.
SOURCE_X, SOURCE_Y, SOURCE_HT	The X,Y location of the source (see COORD_SCALE) and its elevation (see HT_SCALE).
SOURCE_DEP	The elevation of the shotpoint, and the shot depth (see HT_SCALE).
REC_X, REC_Y, REC_HT	The X, Y location of the receiver (see COORD_SCALE) and its elevation (see HT_SCALE).
AZIMUTH	Source-receiver azimuth in tenths of a degree.
SHOT_PEG, REC_PEG	The source and receiver peg (or station) numbers.
TRTYPE	Set to 1 for live data, 2 for dead traces.
DATA_START	Set to the maximum sample time for dead traces.

The processing flow to do this is quite simple.

- **On the Launcher, go to the 'Flows' tab**
- **Click on the 'Job files, button**
- **Select the processing flow 01_merge_geometry.job**



The processing flow 01_merge_geometry.job alongside the ADDGEOM module used to update the seismic trace headers with information from the geometry database.

- **Run the processing flow**
- **Check for any errors in the command window when the job completes**
- **Use the support files table displayed in the logfile to check that the correct geometry, statics and other support files have been used.**

6.2 Checking the Geometry is Correct in SV

It is important to verify that the geometry has been correctly added into the data before we go any further. While mistakes are rare on modern datasets where manual surveying and updating of files has been used errors are not uncommon.

- **On the Launcher, go to the ‘Seismic Data’ tab**
- **Start SeisCat (if it is not already running)**
- **Select the project for this tutorial in the Project pane**
- **Right click, and select “Refresh file listing”**
- **The file pc96-08_geom.shots should now be listed on the Seismic Files pane**
- **Select this file, right click and choose “Open with sv”**
- **Enter op1 for “Output files”**
- **Click OK**

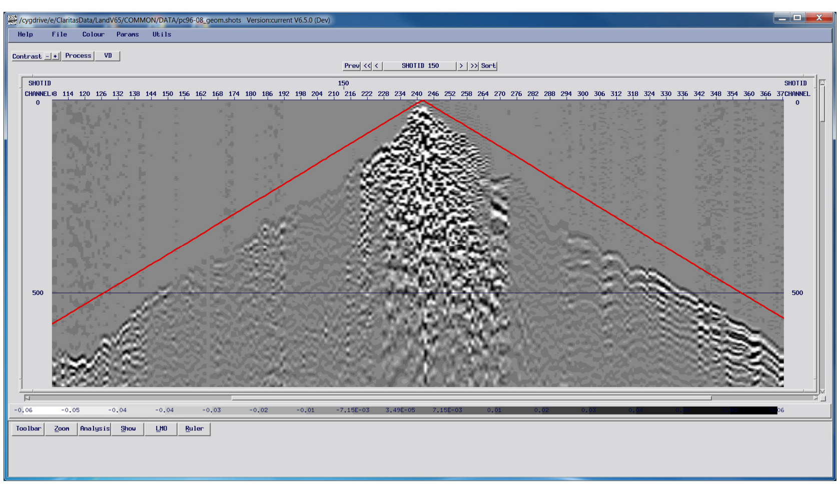
You should now see the data displayed as before.

- **Click on Utils**
- **Select “Overlay offsets”**
- **Supply the velocity as 2300 m/s**

The offset overlay is an effective quality control (QC) check of the geometry. The pattern we see in the seismic data – allowing for elevation and near surface velocities – should match the pattern in the offset. Critically, the trace that has the smallest “first break” time should be closest to zero offset.

Step through the data, confirming this is correct.

If the apex of the red line (defined by the offsets) didn’t match with the first arrival times of the shot record, then the survey information wouldn’t match the seismic. Back-tracking the problem can be challenging, but is often an error in the “relations” (X) SPS file which maps the survey pegs to the file and channel numbers.



Geometry Quality Control (QC) check; the red line shows the offset, converted to a time value using a constant velocity. The first arrivals in the seismic shot match the apex of the red line, showing that the earliest arrivals correspond to the smallest offsets.

You can also use the **LMO button** to apply a "linear moveout" to the seismic traces with a similar effect.

It can also be useful to add the CDP and OFFSET to the display.

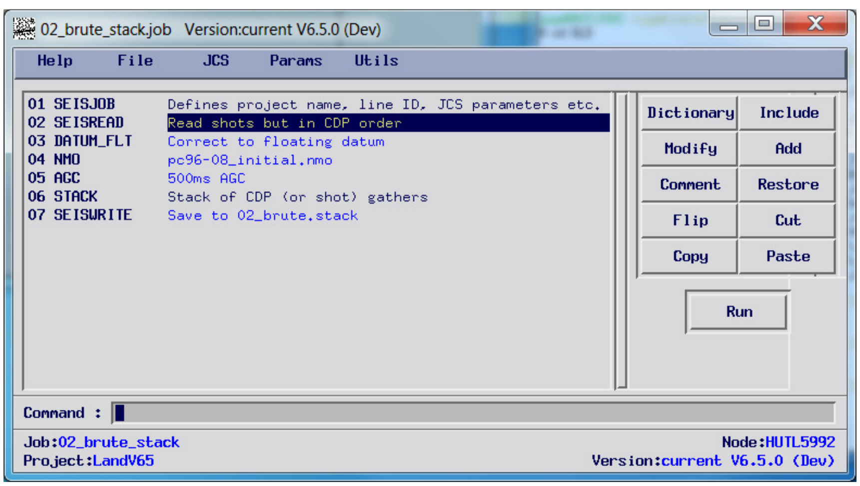
- **Click on Params**
- **Select "Main display"**
- **Use the "Header index" and "Header frequencies" options**

Note that OFFSETS will be displayed in decimetres, and the offsets are "signed" with positive and negative values depending on whether they are "behind" the shot or "in front" of it.

7 Building a Brute Stack

With the geometry in place we can now build a basic "brute stack". A brute stack has little to no processing applied and a very simple velocity model, but serves as a "baseline" for any future processing efforts.

- **On the Launcher, select the Flows tab**
- **Click on "Job files", and load the flow 02_brute_stack.job**



The processing flow 02_brute_stack.job, which creates a basic stacked image of the line.

The HDF5 file format used by ***GLOBE Claritas*** allows you to "sort on the fly" to a new data order, using up to three levels of sorting, each keyed off a different trace header. In this example SEISREAD is loading the data sorted by CDP and CDPTRACE numbers assigned as part of the geometry.

We're applying the **floating datum**, and then using the Normal Moveout (NMO) Correction to "flatten" the hyperbolic reflections in the gather, to remove the effects of offset on the trace curvature.

The NMO correction is using a simple brute velocity function; we'll explore where this came from later.

There's a pre-stack AGC being applied – as we expect the signal to noise ratio to be quite poor, and then the data is summed (stacked) and written out with SEISWRITE.

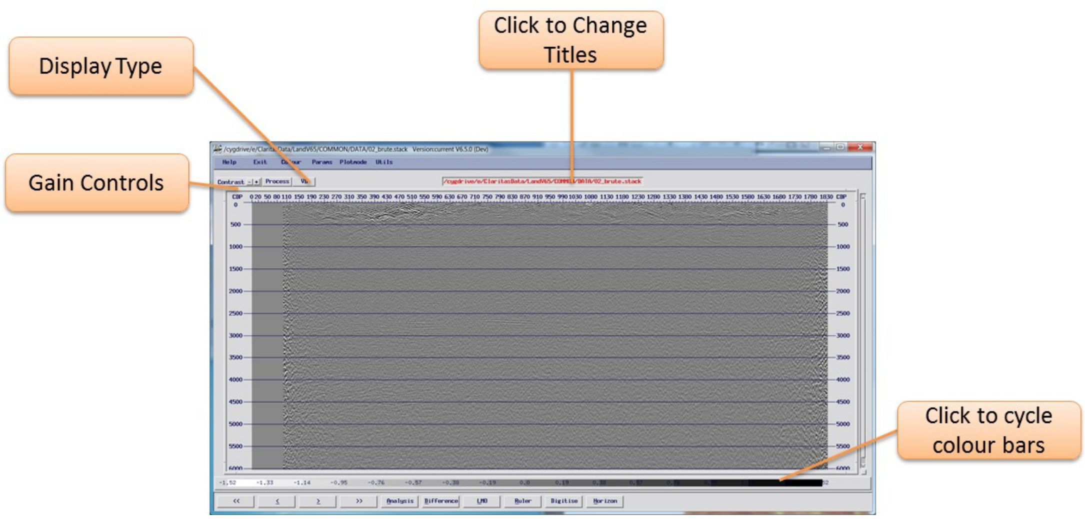
Geophysics Comments:

- The stacked image we create is a "normal ray" or "zero offset" section
- The NMO correction simulates the source and receiver being co-located, but makes no allowances for the dip of events or refraction
- As a result we have a "normal ray" section; the ray-path at any given CDP makes a normal with the sub-surface events; this is a vertical ray only for plane, horizontal layers
- The section is an image of the reflected and scattered wavefield, which means it also contains diffractions off steep dips or discontinuities

- **Run the job 02_brute_stack.job**
- **In SeisCat, select the project then right click and select "Refresh file listing"**
- **Select the file 02_brute.stack**
- **Right click on the file, and select "View with seisview"**

Seisview is another seismic viewing application. Its designed to allow you to view and compare different versions of the same seismic datasets – pre-stack or post-stack – quickly and easily.

Allow the parameters to default, and view the stack.



The seisview display of the brute stack, showing some of the interactive controls.

While there's not much to see of the brute stack, it's worth playing around with some of the display options.

- **Click on the title to change it**
- **Click on the colour bar to change, or right-click to selection options**
- **Click on the Analysis button, select 'Amplitude histogram'**

- **Use the left mouse button to select a region (click and drag) on the seismic display**
- **Use the +/- contrast button to vary how the colours are mapped to the amplitudes**
- **Click on the Process button and enter AGC for the processing sequence as we did in SV**

8 Managing Near Surface Variations: Refraction Statics

8.1 What are Refractions Statics?

On land seismic surveys, we can expect the near surface terrain to vary significantly. In a few meters we might go from soft, marshy ground to a weathered outcrop. This can lead to large, localised velocity variations which distort the seismic travel times.

We can use the same "static shift" approach to correct for this as we did with the variation in elevation, but to do that we need to estimate the near surface velocity variation and replace it with a constant – the same "replacement velocity" we used in calculating the floating datum.

One approach to estimating this velocity is to model it from the direct and refracted arrival; this will give us an estimate of the low velocity layer and its thickness. We can then develop a static shift model to correct for this layer.

We will be able to tell if we are doing a good job because when we combine the elevation and near-surface low velocity layer corrections, the direct and refracted arrivals from the low velocity layer will be linear, without the variations we saw earlier.

8.2 Picking First Breaks in SV – Manual Picking

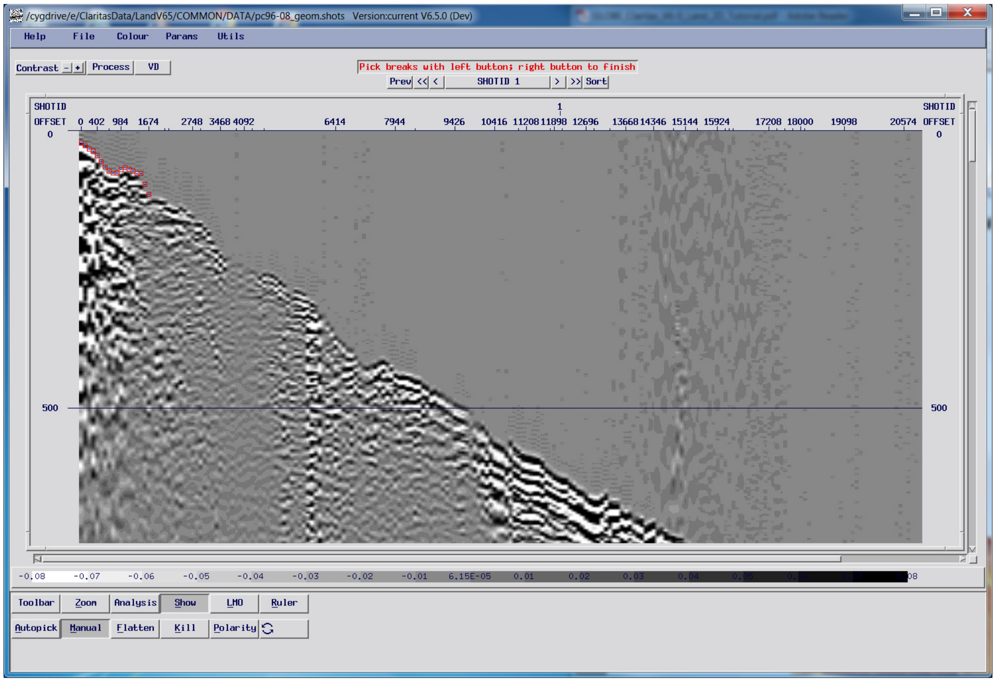
In order to model the near surface velocities, we need pick the arrival times for the direct and refracted arrivals. We do this by picking the "first break" – the start of the first coherent signal on a given seismic shot and channel.

- **In SeisCat, select the file pc96-08_geom.shots**
- **Right click and select 'Open with sv'**
- **Retain the name op1 for 'Output files' and click OK**
- **Click on Toolbar button and select 'Add First break picking buttons' option**

The First Break picking toolbar in SV

You can manually pick the first breaks using the "Manual" button.

- **Click the Manual button**
- **Select OK to ignore the pop-up menu for now**
- **Start picking with the left mouse button**
- **Click once to place a pick, or hold down the button and drag to create a line of picks**
- **Aim to pick the first breaks on the data**



Manually picked first breaks – the red squares – on the first shot record in PC96-08

You can re-pick any trace just by clicking in a new position. Clicking on the "circulate" button  activates "delete mode", which clicking will remove picks one at a time, or click-and-drag to remove a range.

You can "fine tune" picks by using the "Flatten" button, which shifts the data by the pick time and to a constant reference datum. Don't forget to use the +, -, <, > keys on the keyboard to zoom in and out.

You can also pick inside a "Zoom" window – click on the "Analysis" button and select "Seismic data Zoom."

If you spot other problems while picking, you can use the "Kill" button to flag a bad trace for editing, or the "Polarity" button to flip the polarity of a channel.

With all the toolbars, the underlined letter acts as a keyboard "hotkey" to activate that function, minimising the need for mouse movements.

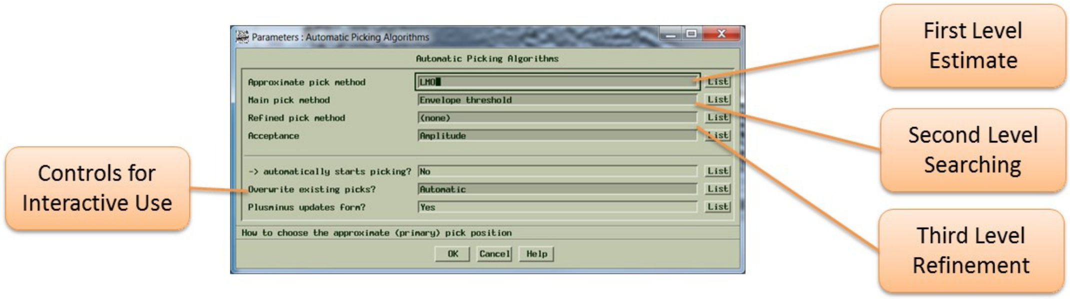
Manual picking is time consuming. It is best left for difficult areas where automatic picking cannot resolve the first break time.

8.3 Picking First Breaks in SV – Automatic Picking

SV has a powerful automatic picking mode.

You can "build" an algorithm using a three step approach with an optional automated check, developing the parameters for each interactively. Once you are happy with the parameters, you can apply them to the whole line.

- **Click on the Autopick button (or press A)**



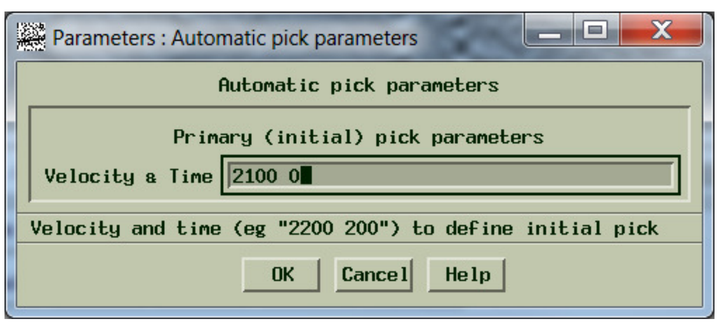
The Automatic Picking parameter form; you can configure a “three layer” algorithm as well as the options for semi-automatic (interactive) testing.

The top four parameters set up how the picking algorithm will be “built”; this is based on an approximate estimate to get close to the first break (in this case a constant linear-moveout or LMO velocity), a “searching tool” that tries to identify the first break, and optional tools to refine the pick.

The bottom part of the form controls the semi-automatic testing; setting the “Plusminus updates form?” parameter to Yes allow us to use the same + and – buttons we used previously to change the display contrast, to also modify the picking parameters interactively.

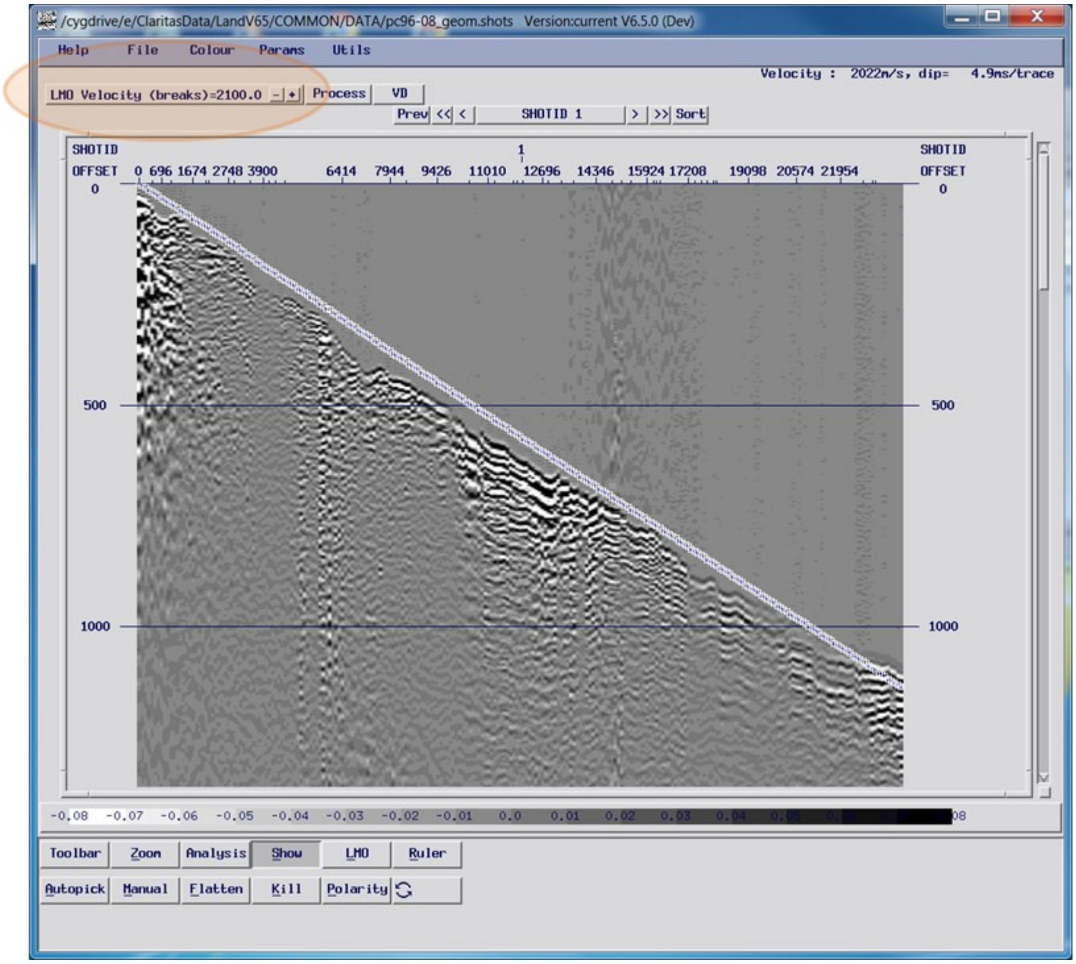
To start with, just select LMO, and set the other pick methods to “(none)”.

When you click on OK, you’ll then be asked for an LMO velocity and a time; specify the velocity as 2100 and the time as 0.



Selecting the initial LMO parameters for Automatic First Break Picking

At this point we’re only after an estimate of the best LMO velocity to use. If you right click on the “Contrast” button you can now select other parameters (eg LMO velocity and LMO T0); the button name changes to show the parameter we are varying, and the +/- controls now change this parameter.



Using the interactive controls to vary the LMO velocity for first break picking. The picks (white) change as the +/- buttons (circled) are used to vary the velocity

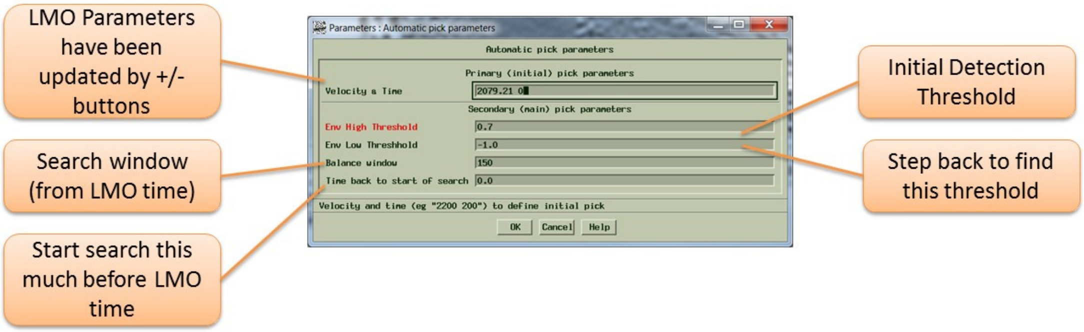
We're going to use "Envelope Threshold" as the main picking routine; this works by searching in a window (from the initial estimate) for the point at which the "envelope" of the amplitudes exceeds a threshold value.

You can add more sensitivity by setting the algorithm to then search up the trace to a lower threshold point – which is useful if the trace is noisy.

You can also tell it to start searching the trace before the time in the initial estimate (ie from the LMO calculation)

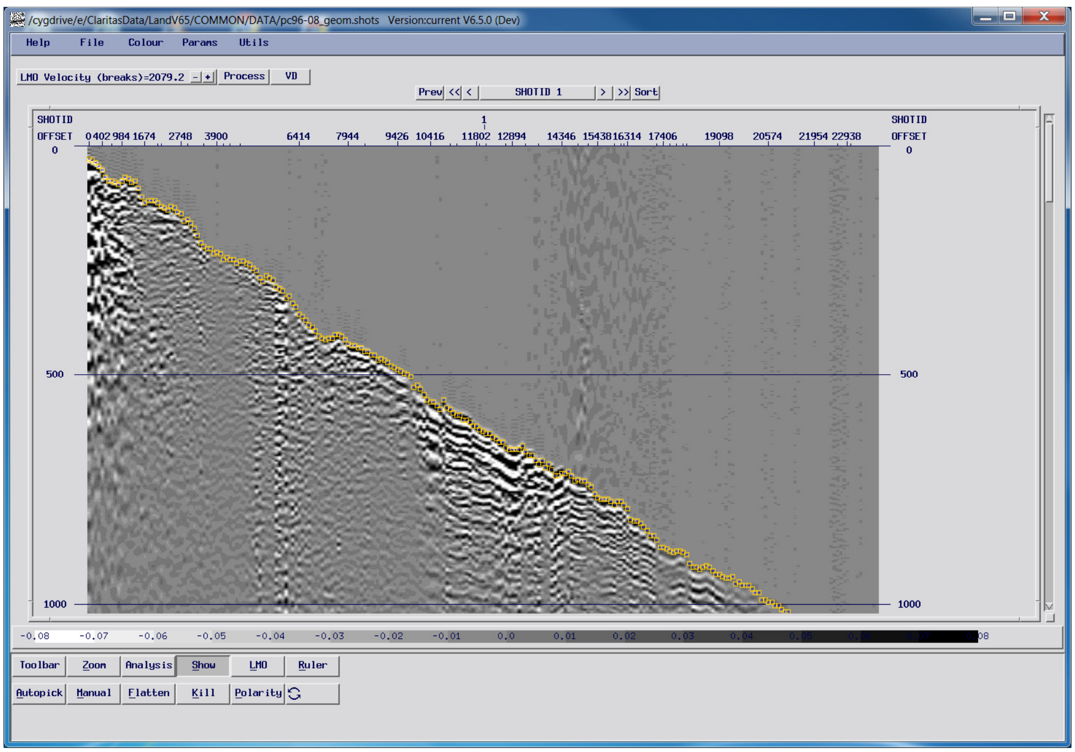
- **Middle Click on the Autopick Button**
- **Select "Envelope threshold" for the Main Pick method**
- **Click OK**

The 'Automatic pick parameters' form will then appear. This will have updated the 'Primary pick parameters' to match any changes to the LMO parameters, made from the interactive tuning with the +/- buttons. The parameters for the secondary pick have also been added to the form.



The 'Automatic pick parameters' form has updated with LMO parameters from our interactive tuning, and has the parameters controlling the "Envelope Threshold" detection method added.

When you click on OK, the shot will then autopick with these parameters; yellow squares will replace the white ones, as we set the automatic picks to overwrite.



Autopicks of the first break (yellow) using LMO as a first estimate, and envelope threshold as a main picking approach

This looks pretty good at first, however we will still need to "tune" the parameters. As before, you can select the parameter to investigate next to the [–][+] buttons, using the right mouse button.

- **If the picks are below the first break, increase the Envelope Time-back parameter**
- **If most picks are above the first break, decrease the LMO parameter**
- **If some picks are being made on noise, increase the Envelope high threshold**
- **To fine tune, press FLATTEN and decrease the Envelope High threshold**

Step through a few shots, verifying the parameters. Jump to shot 142 (by clicking on the SHOTID label) which is noisy, and experiment there as well.

We might not be able to automatically pick very noisy data, so we can use the "Acceptance" option to set amplitude-based thresholds for keeping or rejecting a first break pick.

- **Middle click on the "Autopick" button and set "Acceptance" to "Amplitude"**
- **Click OK**
- **On the second form (Automatic pick parameters) accept the defaults by clicking OK**

The amplitude option searches within a window and looks at the ratio of the pick amplitude to the RMS amplitude in the window. An amplitude ratio of 0.3 works well; you can adjust this parameter interactively as well.

8.4 Autopicking all the Shots

Once you have the parameters configured you can autopick all of the shots:

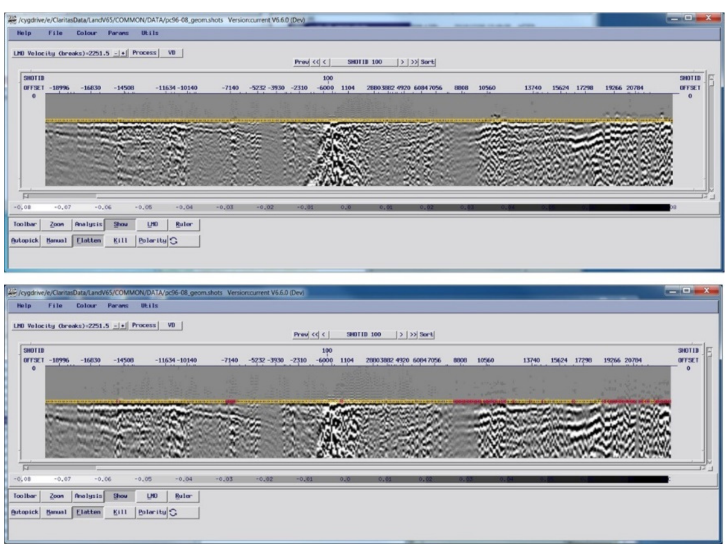
- **Click on Utils on the toolbar at the top of the screen, and Select 'Autopick ALL Shots'**

If your input dataset is in hdf5 format then as you manually pick or run the automatic picking the first break picks are saved into the trace header of the seismic data file. Claritas still saves out the .pic file for both HDF5 and CSEGY datasets, and you can also manually save them (File-Save).

8.5 Checking First Break Picks

To check the automatic first break picks, there's a few things you can do.

The first is to use the "Flatten" button, along with the "Manual" button (F and M keys), to adjust the picks as needed. This view can allow you to make fine-scale adjustments based on the overall waveform shape of the direct and refracted arrivals.



Checking the first breaks and manually editing; in this case the red picks have been altered where the automatic picking is not quite accurate.

When you are happy with the picks, you can click on File (on the toolbar at the top of the screen), then select Exit. The picks are saved automatically.

8.6 Creating Near Surface Statics from First Breaks: Introducing Refstat2D

We use the Refstat2D application to turn the first break pick times into a near-surface velocity model, and then use that model to calculate static shifts.

Refstat2D starts off with an initial model based on the geometry information. The application then ray-traces using the source and receiver locations to come up with theoretical first arrival times; these are compared to the actual arrival times and the model perturbed to minimise the difference between the two.

In Refstat2D we work iteratively, usually allowing the inversion to run through a few cycles. We can then check the model and allow the parameters to vary.

In general, we do this by increasing the range over which the velocities in a layer can vary, or allowing more complexity in the depth or velocity structure.

The goal is to minimise the root-mean-square (RMS) variation between the observed first break picks and the modelled first arrival times. This is displayed as a graph for each iteration, so you can ensure you are making progress.

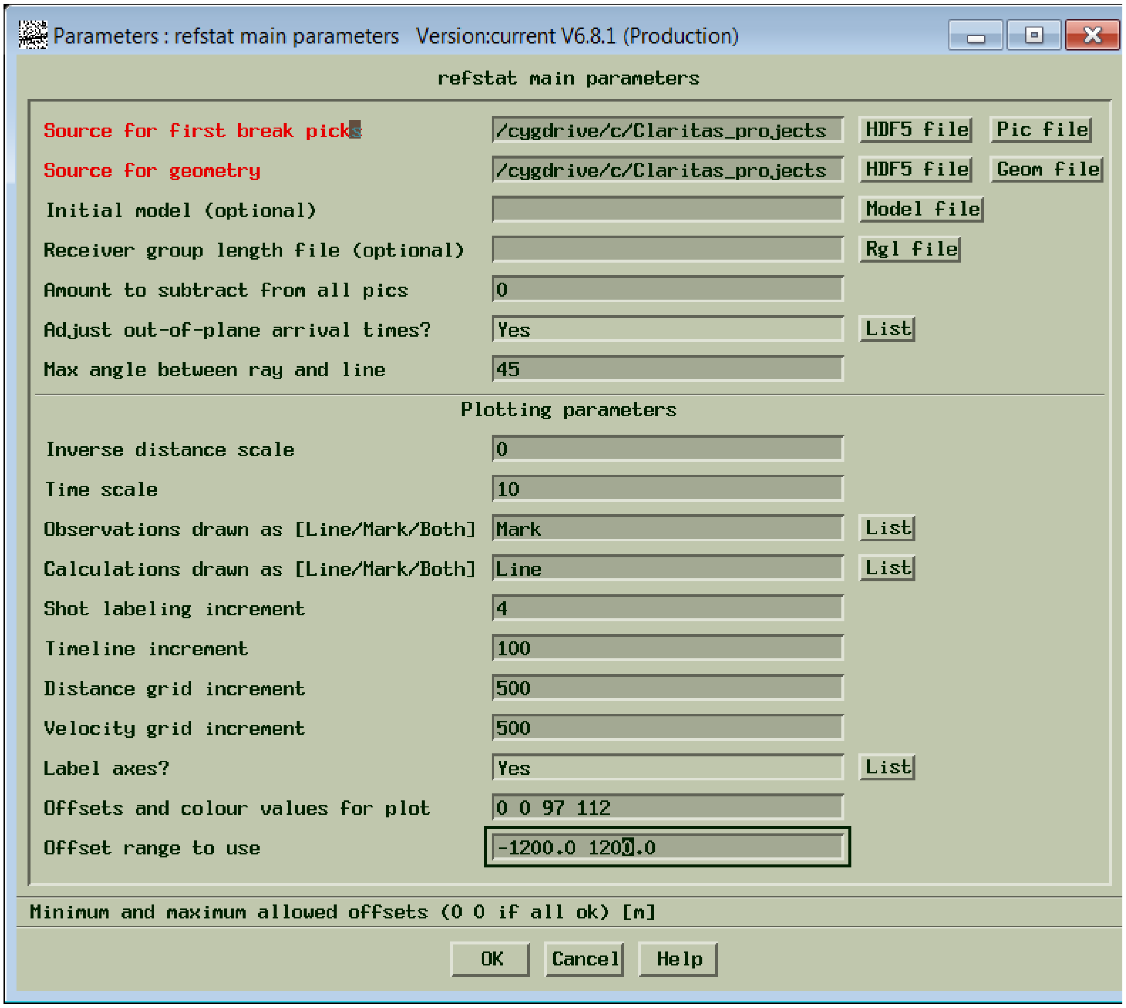
8.7 Getting Started with Refstat2D

On the Launcher, go to the Statics tab and click on 2D refstat.

In the initial form, you will need to select the file containing the first break picks you have made, as well as the file containing the geometry information. The geometry is needed so that the offsets, elevations and shot depths (if appropriate) are correctly modelled.

We can also restrict the offsets we are going to use for the first break analysis. Very long offsets will effectively model deeper layers, and in this case we're interested only in modelling the weathering and sub-weathering layers, so we will limit the offsets to +/- 1200 metres.

The initial parameter form will look something like this:



Refstat2D initial parameter form; note how the offsets are limited in the final parameter

As part of the V6.7 release Refstat2D (and 3D) were improved to be able to read the Geometry and first break picks from the HDF5 seismic trace header (Reading from *.geom and *.pic files is still supported), as such your best set of first break picks will always be selected if using the HDF5 file as input.

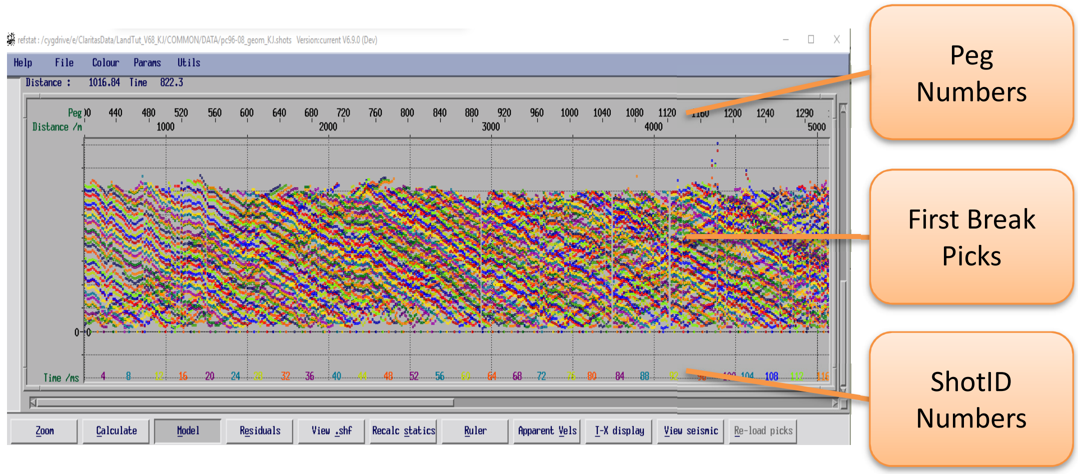
Implemented as a command line option for Refstat2D in V6.8 (will be default behaviour for V6.9) is the -allow_reload option (command line : refstat -allow_reload) which links the Refstat2D application with the SV first break picking tool, allowing the user to update picks whilst developing an optimal refraction static solution.

8.8 Initial Screen Displays

Refstat2D opens up two new windows. The first is the main window that shows a coloured display of the first break picks, and the second is a display showing the current velocity model.

The main window looks complicated, because we are seeing all the first break picks at once. At the bottom of the display are the SHOTID numbers, colour coded. The first break picks at a given shot are the same colour as the SHOTID number, with the colours cycling through in sequence.

The vertical axis is the first break time, and the peg numbers (for shot and receiver locations) are labelled at the top of the display.



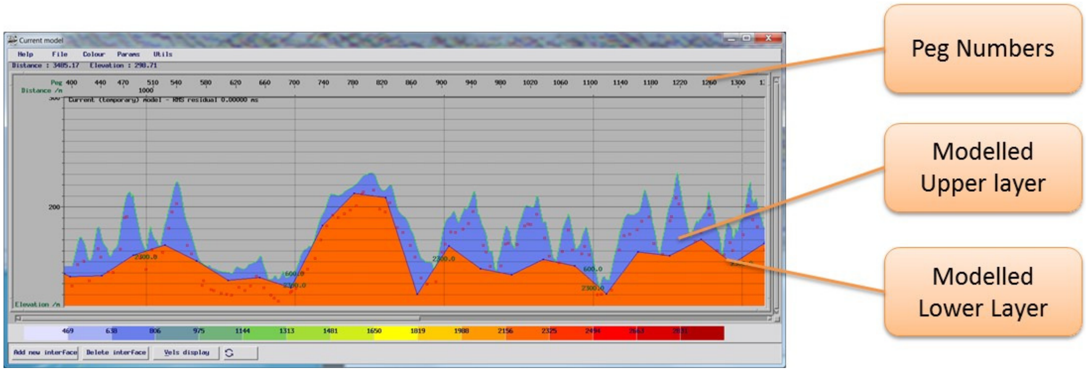
The Main Refstat Window. The first break picks are shown colour coded by their SHOTID number, which is labelled along the bottom. The Peg numbers are at the top. Time (of the first break pick) is the vertical axis.

If you click on the Main Display the closest shot to the mouse pointer will be colour coded white – you can identify all of the first break picks associated with that shot, displayed as a function of their receiver peg location and time.

If you read in the geometry and picks from an hdf5 file you will see two additional buttons on the Main Refstat window. The first of these buttons (**View seismic**) allows the user to launch the SV application using the input HDF5 dataset, used to define the geometry and first breaks information used by Refstat. SV opens in first break picking mode with the existing picks from the trace header displayed. Users can the use the Residuals display in conjunction with the SV application to identify and update poor quality picks. The Refstat application monitors the input HDF5 file and identifies when it has been updated, when this happens the **Re-load picks** button becomes live and the user can choose to load these updates for use in the analysis process.

As with other GLOBE Claritas displays you can use the **+**, **-**, **<**, **>** keys to change the display scale quickly; don't forget to use the Shift key where appropriate to reach the correct symbols.

The Model Window shows the current velocity model, with a structural representation of the sub-surface layers in depth. The colour of the layer is the current velocity assigned to that layer.



The Model Window. The sub-surface structure is shown at depth, with the layers coloured based on their velocity. Shot locations are red squares.

Initially we have two layers; the first layer tracks the elevation topography for its top, and has a smoothed version of the topography for its base. The layers are colour-coded to show their velocity. The shot locations are displayed as red squares; in this case the shots are show at their shot-depth. In this initial, very simple model, some of the shots are below the weathering layer, and some are inside it.

The layer structure is controlled by “control points”; these are the blue dots on the interface.

8.9 The Sub Weathering Layer

The first thing we need to do is to update the “sub-weathering layer”, or the first layer in the velocity model. This represents the immediate near surface, and we’re going to estimate this from the gradient of the first breaks across the first few hundred metres.

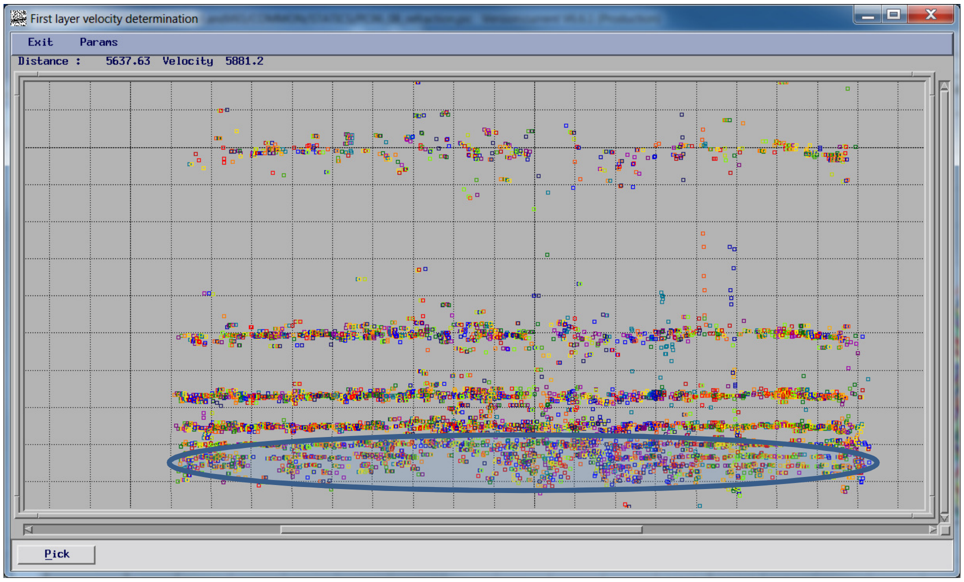
This is the “apparent velocity” of those first breaks – just the distance divided by the arrival time – and we can estimate this from the data using the **Apparent Vels** button on the main display.

Click the button, and follow the on-screen instructions to set the “bounds” for the analysis.

Define the upper boundary around 200ms, using the left mouse button to pick points, and the right mouse button to finish.

Then define a lower limit at perhaps 50ms in the same way.

When you have finished picking with the right mouse button on the lower limit, you will see a display of the apparent velocities.



The Apparent Velocities display. Velocity is the vertical axis, the low velocity trend is circled.

With the simple linear division process we get "whole sample" rounding errors that produce the "banding" or "clustering" of results, but we can also see the "low velocity" trend we want to pick through between 500m/s and 1000m/s

Click on the pick button, and select "turn picking on", then use the left mouse button to pick a trend through the lowest velocity "point cloud"

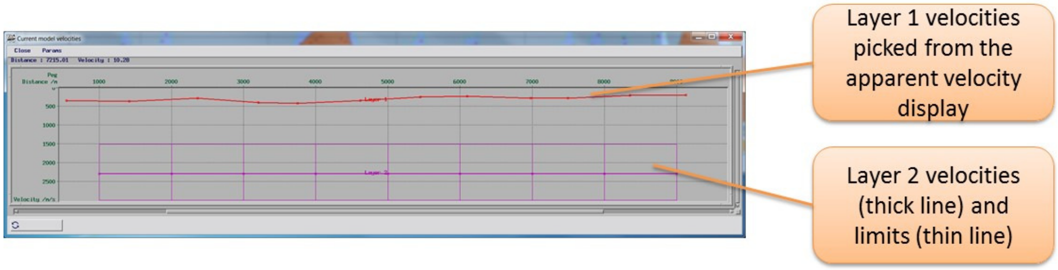
Then, click on the Pick button again and select "modify model with picks" and leave the parameters at their default values. This will update the first layer in the model with the velocity we have picked, and not allow that velocity to vary.

8.10 Modifying the Model

You can manually adjust the structure or velocities in the model. There are buttons to Add or Delete an interface, and if you click on the circulate button  then you can add, double, delete, move or edit the control points on an interface.

Initially, it's a good idea to start with a simple model, and then add complexity as the model converges.

If you click on the "Vels display" button this will open a new window where you can view and adjust the velocity information.



The velocity display window. The velocity in each layer is shown as a thick line, with the bounds that velocity can vary within shown as thin lines. The upper layer (red) has been picked from the apparent velocities display, and fixed so it cannot vary.

The velocity of each layer in the model is shown as a thick line, with dots representing the "control nodes" that allow the velocity to vary within a layer. The Thin lines represent the limits within which the velocity can vary. You can use the circulate button to modify the control points, the entire layer, or the maximum and minimum velocity limits. As with the structural model, it is usually best to add complexity into the model once you have made some initial progress.

8.11 Running the Inversion

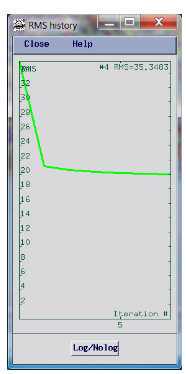
To run the inversion use the "Calculate" button on the main windows display (which shows the first break picks); this will lead you to a form which asks for a name for the output files, and the number of iterations to use.

Provide a name (such as pass1) and set the number of iterations to 6, then click on OK.

As the inversion runs a new window opens. This window shows you the root-mean-square variation between the first break picks you made and the results of the ray-traced inversion for each iteration. With each iteration Refstat2D will adjust the velocity and structure to try and reduce that RMS value.

Typically the RMS might start at around 40, but after six iterations it has dropped to something like 16.

You will also notice that the RMS curve will flatten out, as the model starts to run into the constraints on structure and velocity we have set.

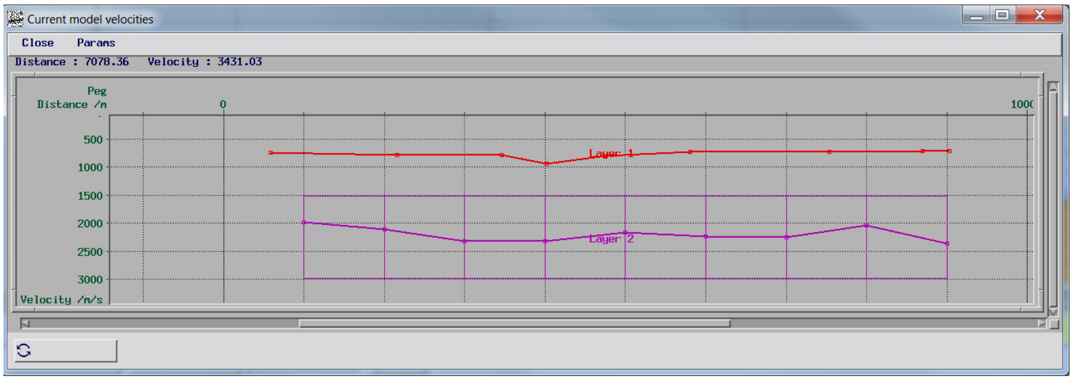


The RMS window for the first 6 iterations; the initial large differences between the picks and model have been reduced with each iteration of the tomographic inversion.

8.12 Adjusting the Model

After running the inversion for a few iterations and reaching a "plateau" in the RMS, we need to modify the model constraints to proceed.

The three things we can do are to:
- Relax the velocity limits on one of the layers
- Add more control points into the velocity model, to allow greater variation
- Add more control points into the structural model, to allow more complex structure



The Velocities window after the first 6 iterations; note how the layer 1 (velocities in red) are fixed, but the velocities in layer 2 (purple) can vary.

8.13 Looking at the Residuals and Statics

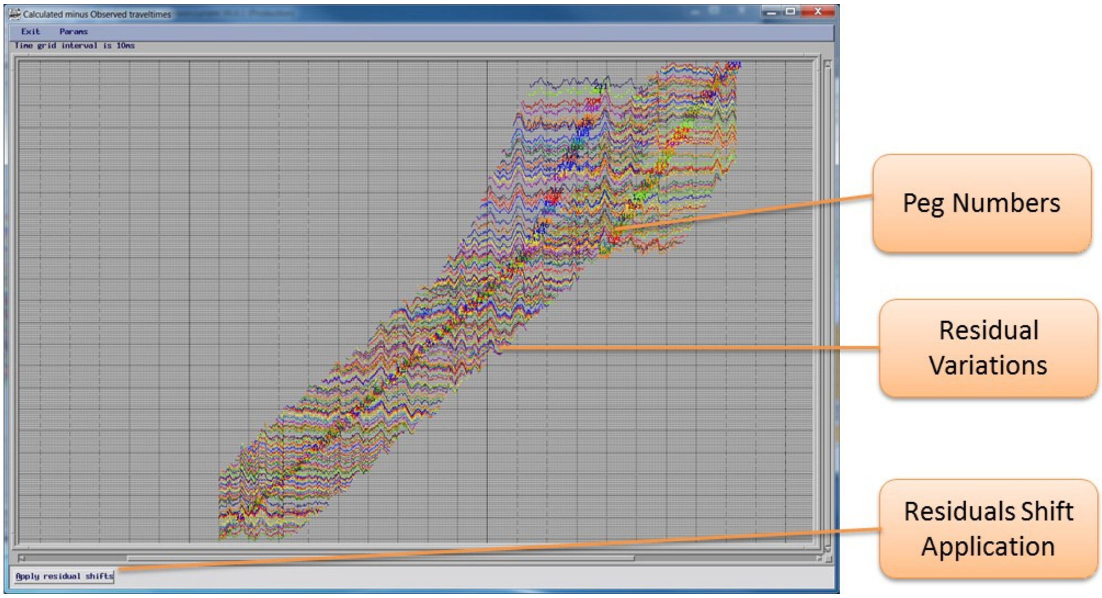
Refstat2D lets us compare the ray-trace results with our original picks. If you go to the main window with the first break picks and click, the picks will be highlighted in white as before, but you will also see a thin, solid line that represents the modelled arrival times.

This can be hard to interpret, but if you press the "Residuals" button then you can look at the differences between the modelled and picked first breaks in more detail.

In this display you can see the differences between the modelled and picked first breaks plotted directly as lines for every shot. A flat line would indicate no variation – an exact match.

By looking at the spatial variation of these residuals we can identify areas where there are local structural or velocity variations below the scale of our current model.

The Refstat tool also breaks these down into "surface consistent" variations associated with a given shot or receiver peg (station) location; in this way the model only needs to capture the "long wavelength" part of the solution; the "short wavelength" variations can be resolved through a surface-consistent process.



The residuals display; flat horizontal lines indicate a "perfect' fit. Where the residuals have a common structure across multiple shots, it indicates a local velocity or structure variation.

Where there's a vertical trend that runs across multiple shots, this shows you where the model is too simple - we need to add complexity in the structure or the velocities. Where there's an anomalous "spike" on a single shot this is showing where we might have a miss-pick on the first breaks or a geometry issue that we have missed.

Clicking on the "**apply residual shifts**" button applies these surface consistent short-wavelength variations. The lines should become much smoother, and much flatter. This is where the -allow_reload functionality should be useful as after the residual shifts are applied any areas where first breaks picks are poor or obviously mis-picked can be analysed by clicking on the View seismic button, then by double clicking on the shot you wish to QC in the residuals display the SV application will navigate to this shot, the first break picks are automatically displayed and you can update as required, when you have finished adjusting your picks in SV and the seismic trace header has been updated the reload picks button will display a warning triangle to alert you that the first breaks should be reloaded to take advantage of the update. When you have made all the updates you feel necessary you can run the inversion again.

You can look at the statics created from the model and the surface consistent solution by clicking on the **View .shf** button on the main Refstat2D display window.

The statics view window. Receiver statics are in blue, with the shot statics in red. The top display shows the long-wavelength statics derived from the model, with the bottom display showing the surface consistent residual solution.

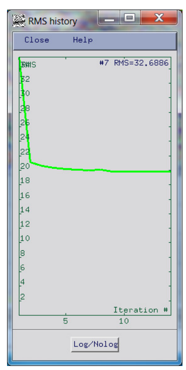
We can see from this that the residual shifts are still quite large – as big as those from the model – and so while we could potentially stop the process here and have some significant improvements, we should probably adjust the model further.

8.14 Adjusting the Model Further

The first place to add complexity in the model is with the velocity structure.

Go back to the Current Velocity Model display and use the circulate button to select "Double", then click on the second layer velocities.

Then click on "Calculate" and run 6 more iterations.



After doubling the velocity control points in layer 2 and running 6 more iterations, the RMS hasn't changed much.

This hasn't made a big different to our solution – the RMS is about the same.

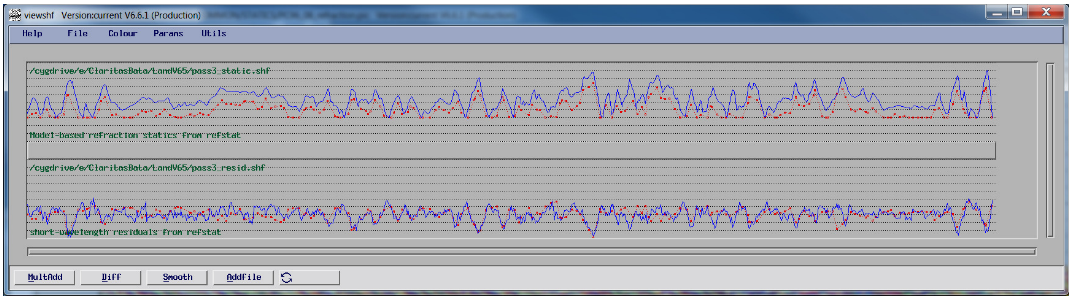
We may need to allow the structure to have more freedom. In the Current Model window, right click on the "circulate" button and select the "Double" option, then click on the model.

This doubles the number of control points on the interface, so that the structure can be more complex.

Run another iteration; in my case this takes the RMS down to about 14 or so, again starting to plateau. Checking the residuals and statics displays as before we can see we are making good progress.

Double the points on the second layer again, and continue.

This takes the RMS error value down to around 8 or 9. Check the residuals and statics as before.



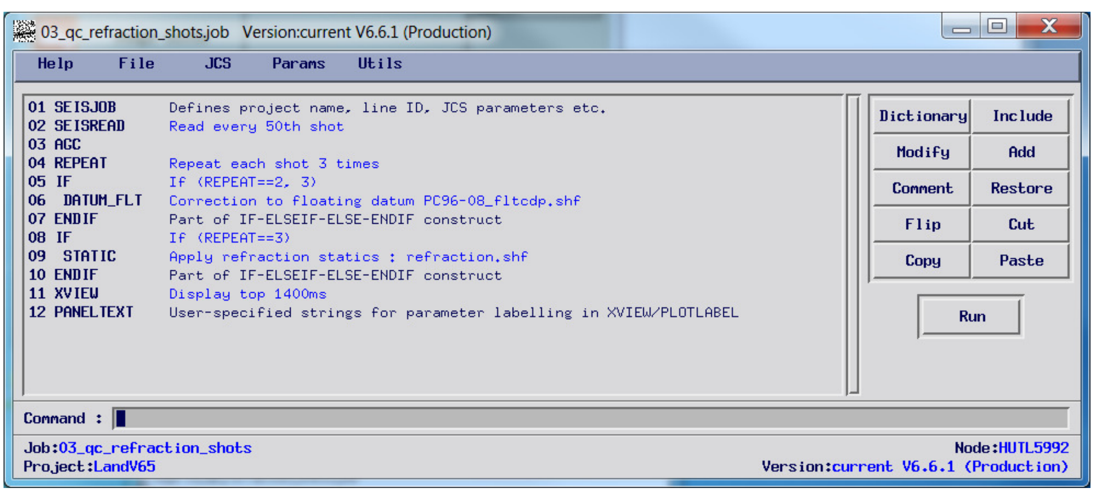
The model based (top) and surface consistent residual (below) statics after three rounds of iterations; the model based statics are now larger than the residuals.

We could continue to iterate, using the surface consistent residuals as a guide to where we can add more complexity into the structural model, however this is probably sufficient.

As well as displaying the long- wavelength (from the model) and short-wavelength (from the residuals) refraction statics solutions the viewshf window also plots the combination of these. All three static correction plots will be automatically saved in the STATICS folder as *_static.shf, *_resid.shf, and *_total.shf.

8.15 Applying Refraction Statics and Checking the Result

The processing flow **03_qc_refraction_shots.job** can be used to look at the impact of applying the correction to the floating datum and the combined short- and long- wavelength refraction statics solution.



Job flow to apply the floating datum corrections (for rugged topography) and the refraction statics solution we have generated from the first break picks.

This flow demonstrates some useful techniques for checking datasets or processing sequences.

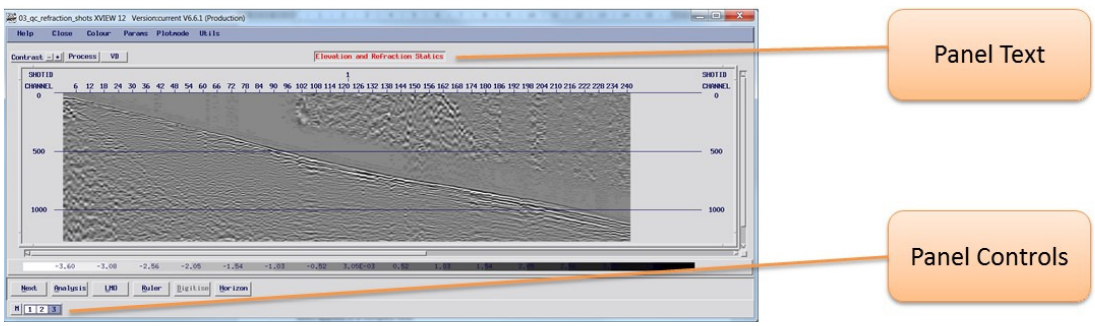
SEISREAD is configured to read in a selection of data, in this case every 50th shot.

REPEAT is used to make three copies of each shot; it also sets up the trace header REPEAT to be numbered from 1 to 3 for each of the copies

IF/ENDIF can be used to make selections within a processing flow; in this case we apply the floating datum correction to REPEAT copies 2 and 3, but the refraction statics are only applied to repeat copy number 3.

PANELTEXT can be used to create panel labels for the XVIEW display so we can easily see what has been applied in a complex flow.

When you run the job, XVIEW automatically configures to have a different panel for each of the repeats, indicated by the numbered boxes lower left. You can move between panels using the mouse, by typing the number of the panel on the keyboard or using the arrow keys.



The panel display produced by the job 03_qc_refraction_shots.job; the panel controls can be used to toggle between the different panels with the left mouse button – but you can also type the number of the panel or use the left and right arrow keys.

When you move between panels you should see the direct and refracted arrivals becoming less complex and more linear; some of that will be from the floating datum correction, but the refraction statics solution should also help a lot.

Note that with XVIEW within a processing flow like this you can only step forwards through the data (with the NEXT button); you cannot go backwards without rerunning the processing flow.

The processing flow **04_refract_stack.job** creates a stack with these two static shifts (the floating datum and the refraction statics correction for the near surface) applied.

Run this job.

Now use SeisCat and compare this stack to the brute stack we created earlier; remember to right click on the project name to refresh the file list first. Use 'Ctrl' click to select both stack files, right click and select 'View with seisview'.

Even with the same brute velocity file as before, and no signal processing applied, there should be a significant improvement in the cohesion and quality of the seismic reflectors in the stacks.

9 Shot-based Pre-processing and Noise Suppression

9.1 Goals of Pre-processing

There are three basic things we want to achieve with pre-processing the data:

- we want to compensate for acquisition related effects
- we want to remove random noise from the data
- we want to remove unwanted (ie non-reflection) coherent energy

This will make it much easier to identify seismic reflectors, which in turn will help us to produce a better image of the sub-surface.

9.2 Amplitude Recovery Tests

The seismic signal loses energy through a combination of spherical spreading and inelastic processes within the earth. We can compensate for this by applying different analytical functions – such as a spherical divergence correction or time-varying gains – or using some form of trace normalisation.

The processing flow **05_gain_panels.job** is designed to test and compare different types of pre-stack amplitude recovery.

Some key things to note in this flow are:

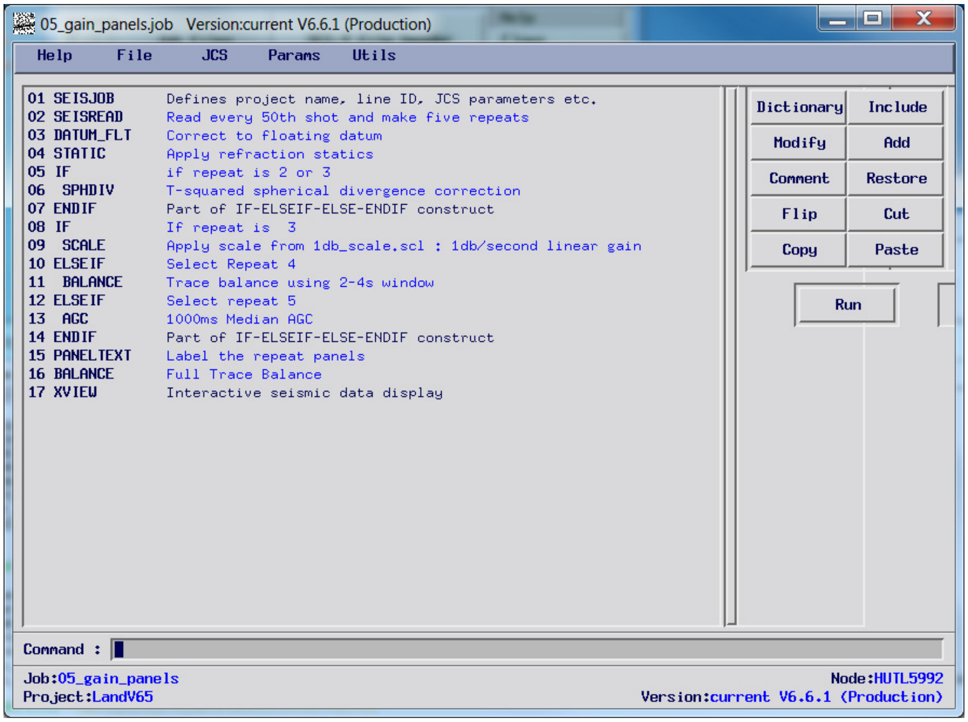
SEISREAD has been configured to make repeat copies of the selected input data, instead of using the REPEAT module. (NREPEAT parameter in the advanced parameters section of the SEISREAD form).

SCALE uses an external "spreadsheet" file to create a 1dB/second linear gain; while we are applying a constant gain function, this type of file can be used to vary the gain in time and space.

ELSEIF is another function that can be used to control data as it runs through a processing flow.

AGC is used here as a "median" or "robust' automatic gain control; this takes longer to calculate than a standard (mean) gain control but handles large amplitude variations without creating scaling shadows.

The final **BALANCE** (module number 16) serves to ensure all of the trace amplitudes have the same overall RMS amplitude level to make the results easier to compare.



The processing flow 05_gain_panels.job, used to test different forms of amplitude recovery.

When you run the flow you will see five shot records, with five repeat panels labelled by the type of amplitude recovery that has been used.

To review the data use the Amplitude Decay analysis window:

- click on Analysis
- select 'Amplitude decay curves'
- click on one of the shots with the left mouse button
- the amplitude decay analysis window will open
- click on AllPanels – this produces an amplitude decay display for each gain test
- click on SyncPanels – this links the amplitude decay curves to the seismic panels, so these will change together when you flick back and forward between the panels on either display

Geophysics Comments:

- The AGC response will remove much of the amplitude variation from the data; this can prevent amplitudes being used qualitatively or quantitatively during interpretation, and may have an impact on pre-stack migration

9.3 Creating a PowerPoint from XVIEW

You can create a PowerPoint format display of the test panels by going to the Utils menu and selecting 'Produce a PowerPoint presentation'. The created PowerPoint can also be opened by the OfficeLibre package on Linux systems. In the slideshow, there will be one slide created for each of the seismic panels and any associated analysis windows.

9.4 Using AREAL to Look at Signal and Noise

With the amplitude decay compensated for, we can look at the amplitude variations across the whole line. To do this we can use the NOISEQC module to create and populate trace headers holding estimates of the signal, noise, and the signal to noise ratio. We can then view these graphically.

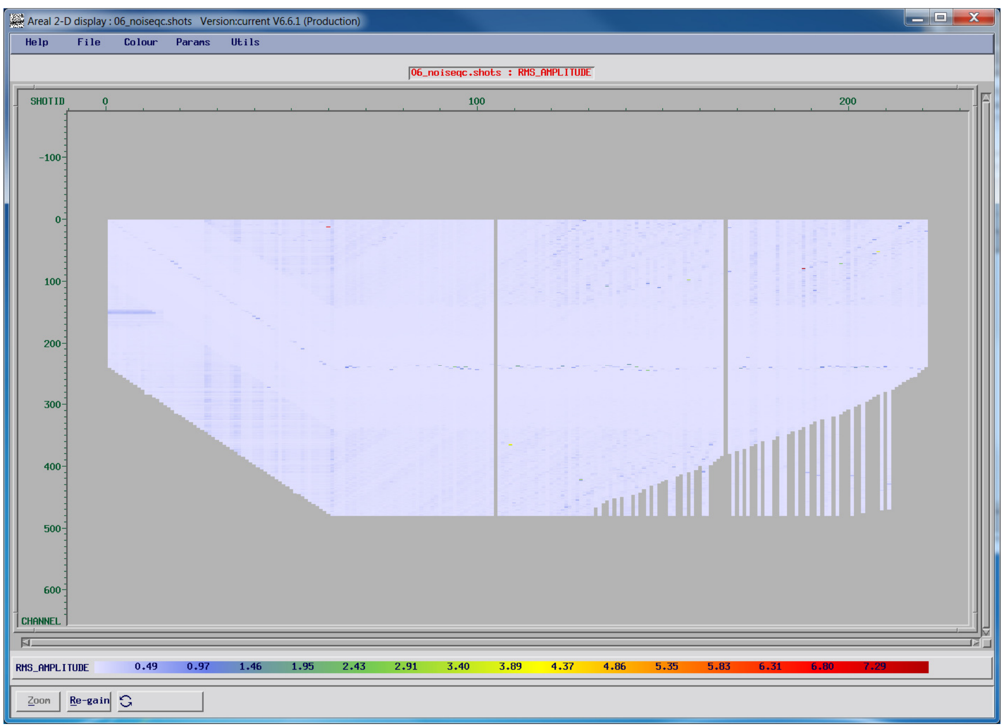
The processing flow **06_noiseqc.job** has been set up to do this, with the third option (spherical divergence and linear gain) selected for amplitude recovery.

Run the job, and then update SeisCat to show the dataset that has been created (**06_noiseqc.shots**)

If you right click on this dataset you can select 'Trace header cross-plot (Areal)' to review the trace headers that have been created graphically.

Select RMS_AMPLITUDE for the 'Display header', and SHOTID/CHANNEL for the 'Axes headers'.

When the panel initially displays, it is scaled to the minimum and maximum amplitudes – that allows us to see any large amplitude noisy traces we might want to remove.



Initial AREAL display of the RMS_AMPLITUDE trace header; there are a small number of high amplitude traces dominating the display.

We can also add the peak amplitude display as well; under the Params menu, select "Add another panel" and choose "PEAK_AMPLITUDE" for the 'Display header'.

If you adjust the 'Areal display parameters' (also under the Params menu) we can then make the display a little bit easier to use; set the Cell Sizes to be the same (in width and height). Then use the +/- keys on the keyboard to zoom in.

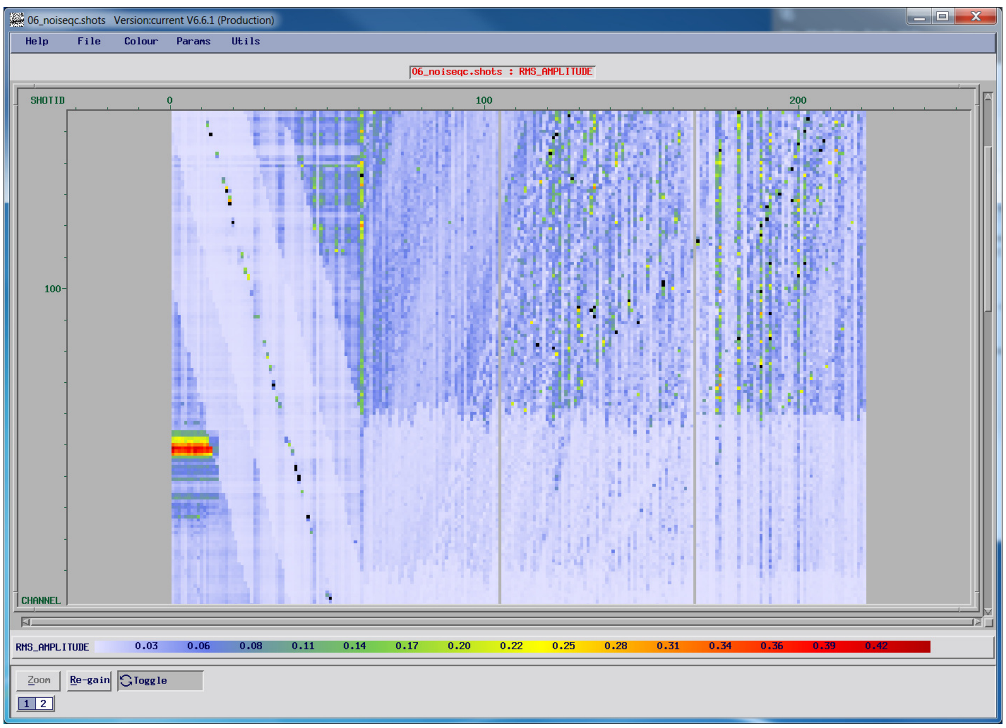
If you click on the "Circulate" button, one of the options is "Set"; when "Set" mode is engaged, you can flag a cell (and its corresponding trace) for editing.

Click on the cells that are red – these have the highest amplitudes – then press the Re-gain button.

When you click on a cell, it puts a back cross on it to show it has been flagged. This can be hard to see, so we can also set this to be a colour (eg black) on the 'Areal display parameters' form.

It can also be helpful to have a separate colour bar for the peak and RMS panels, which is another option on the display parameters form.

While you are editing, the amplitude of the current trace is shown (after 'Value : ' along the top of the plot); this can be useful to compare the current trace to the neighbouring traces - after applying gain corrections traces should never be more than one order of magnitude larger than their neighbours.



The AREAL display part way through editing showing flagged traces (black squares)

The other "Circulate" button controls in AREAL allow you to clear flagged traces, toggle flags on or off, or set entire rows or columns. You can also use the space bar instead of clicking the mouse button.

Use both the PEAK and RMS amplitude to guide your editing; you can also open the seismic display in SV if you are worried about editing out "good" traces by mistake.

When you exit, the trace edits are automatically applied to the seismic file – or you can optionally restore the file back to how it was when you launched areal (options under 'File' on toolbar menu).

9.5 Picking a Refraction Mute

Now that we have addressed the near surface velocity variations – at least in part – the refracted and direct arrivals are no longer useful. In fact the strength of these events can create artefacts when it comes to the application of some signal processing techniques, so we need to pick a mute to remove them.

We can use the same tool to pick the mutes as we used to pick first breaks – SV. You can launch SV directly from SeisCat by selecting the same file we used above (**06_noiseqc.shots**) and right-clicking to open the menu (and bring up the 'Open with sv' option).

Remember to give the output file a suitable name.

You will need to activate the "muting" tool bar – click on 'Toolbar' and then 'Add Muting buttons'

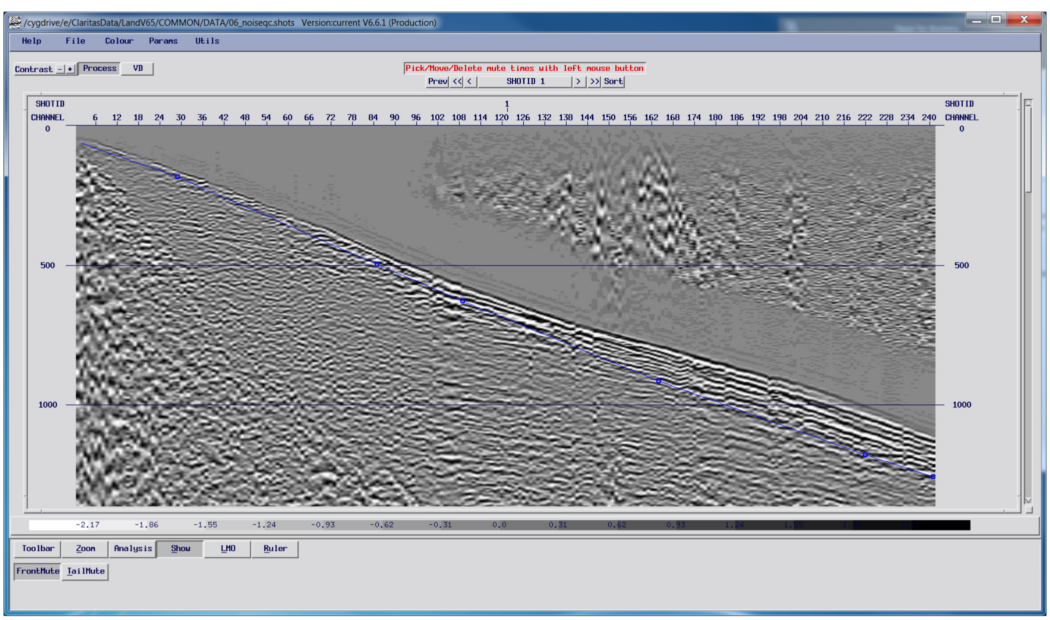
Next, click on the Process button and type "AGC" when asked for a processing sequence; this will make picking the mute a little easier (leave the parameters as default) Use the contrast -/+ buttons to adjust the colour scale if needed.

Now select "Front Mute", and allow the secondary key to default to OFFSET.

Leave the extrapolation mode as Sloped, and leave "Allow picking signed offsets" as "No".

All of the picking is controlled by the left mouse button; the first click makes a pick, and the second click deletes it. You can click and hold to move a pick.

As you pick the mute is automatically replicated on both the positive and negative offsets, and interpolated from the points you have picked. Start picking in the shallow, near offsets and work to the deeper, far offsets.



Picking a refraction mute in SV on the first shot in the line. The picks are blue squares, with the mute automatically interpolated.

Use the << and >> buttons to navigate through the data; the mute is automatically shown as interpolated, so that you can choose to make adjustments – or not – as needed.

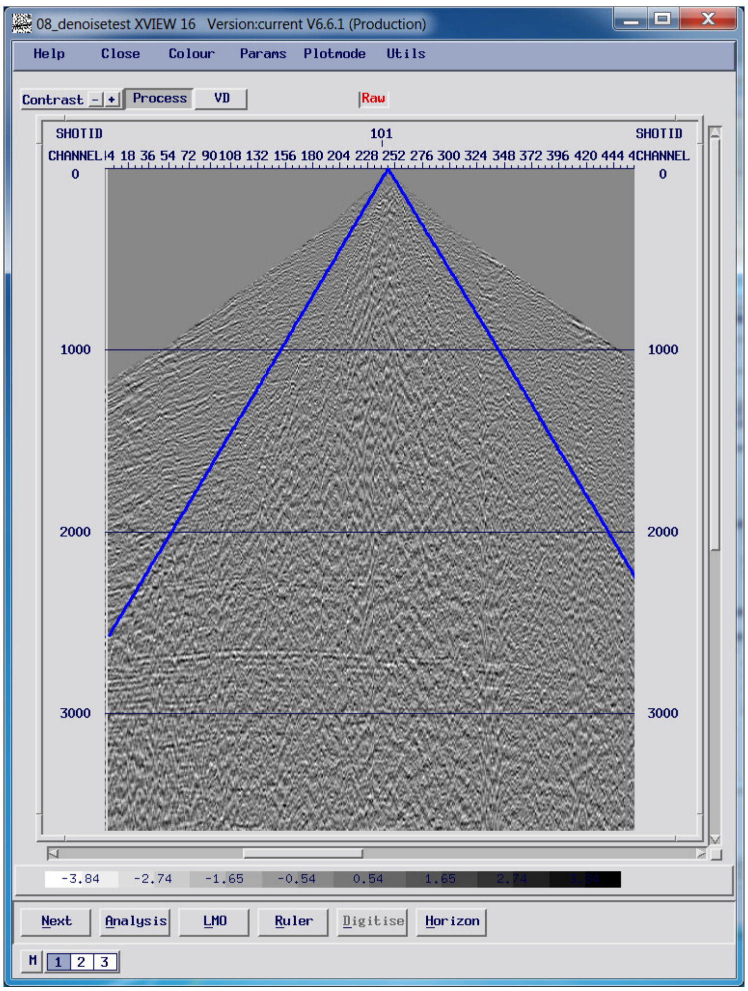
When you have finished (and File-Exit), the mute will automatically be saved in the correct location in the right format.

9.6 Noise Suppression Testing FK and FX

The second type of coherent noise we need to address is the “ground roll cone” clearly visible on all of the shots.

If you review the shots in SV, you can activate a ruler (Ruler button at bottom of plot) with the left mouse button and measure the apparent velocity of this “noise cone” – it arrives at around 850 – 900 m/s.

The noise is steep dip and low frequency, and makes it difficult to see reflected events within this cone.

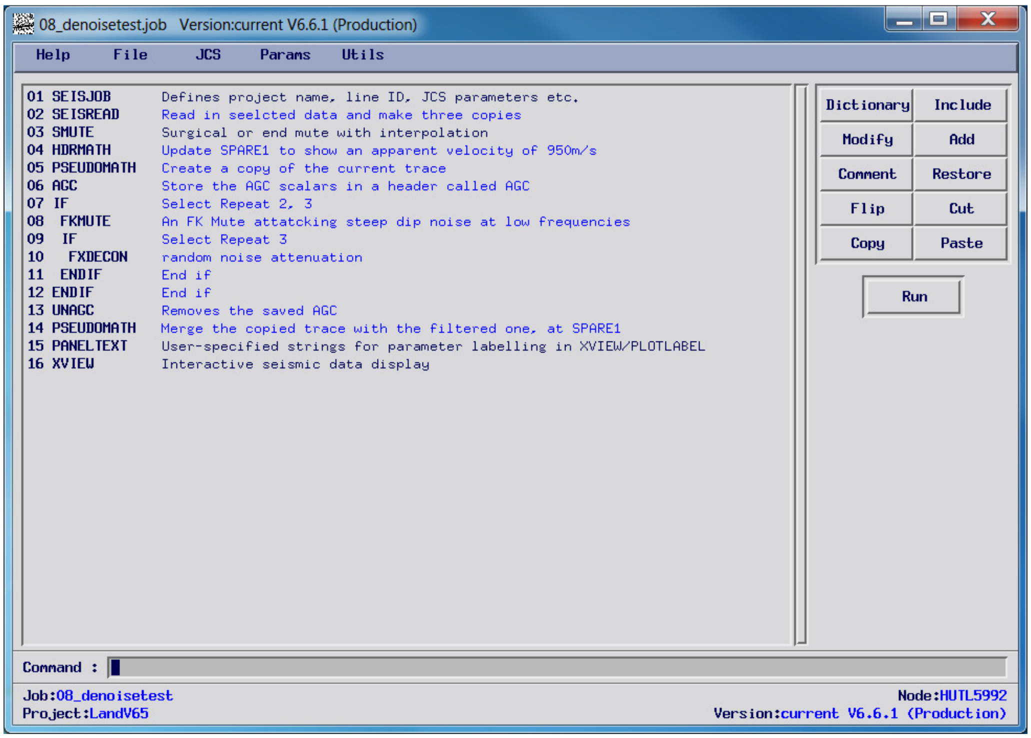


A single shot showing the ground roll “noise cone” below the blue line; this line has an apparent velocity of 950 m/s, and the noise cone is contained below this.

While we could apply noise suppression across the entire shot record, one technique we can use in ***GLOBE Claritas*** is to only attack this noise within that “cone”; in practice we apply a filter to the entire shot, but then merge the filtered shot with the original data along a line that defines this “cone.”

There’s a number of different noise suppression methods we could use. In this example we will employ an FK-domain mute and an FX deconvolution. The FK-domain mute has a few advantages over the standard FK dip filter in that we have more control over the frequency and wavenumber we are targeting.

With land data – or where we have a lot of amplitude variation – it can be useful to use a removable AGC “wrap” around multi-channel processes to avoid artefacts.



Processing flow to test the application of noise removal methods below the “noise cone”

Open the processing flow **07_denoisetest.job** using **XSJE**, and look at how this has been constructed.

SEISREAD is configured to select 5 shot records, and make three “repeat” copies.

SMUTE applies the refraction mute we picked earlier.

HDRMATH can be used to perform mathematical operations on trace headers, using the NUMPY libraries and functions. In this case it is being used to calculate the limits of the “cone” shape, remembering that the offsets in this case are in decimetres. The ABS() function allows for the positive and negative offset variation. The result is stored in the header SPARE1

(See : https://docs.scipy.org/doc/numpy-1.10.0/reference/routines.math.html)

PSEUDOMATH is used to create or manipulate “pseudo-traces”; these are trace headers stored as an array with the same dimensions as the seismic trace. They have a wide range of uses, in this case it is being used to create and store a copy of the original trace.

AGC has been configured to be removable; this means storing the scaling factor for every one of the AGC gates along the trace; this is another example of a pseudo-trace.

FKMUTE applies a mute in the FK domain; this is a more precise approach that using a standard FK dip filter; we’ll look at how this was defined later.

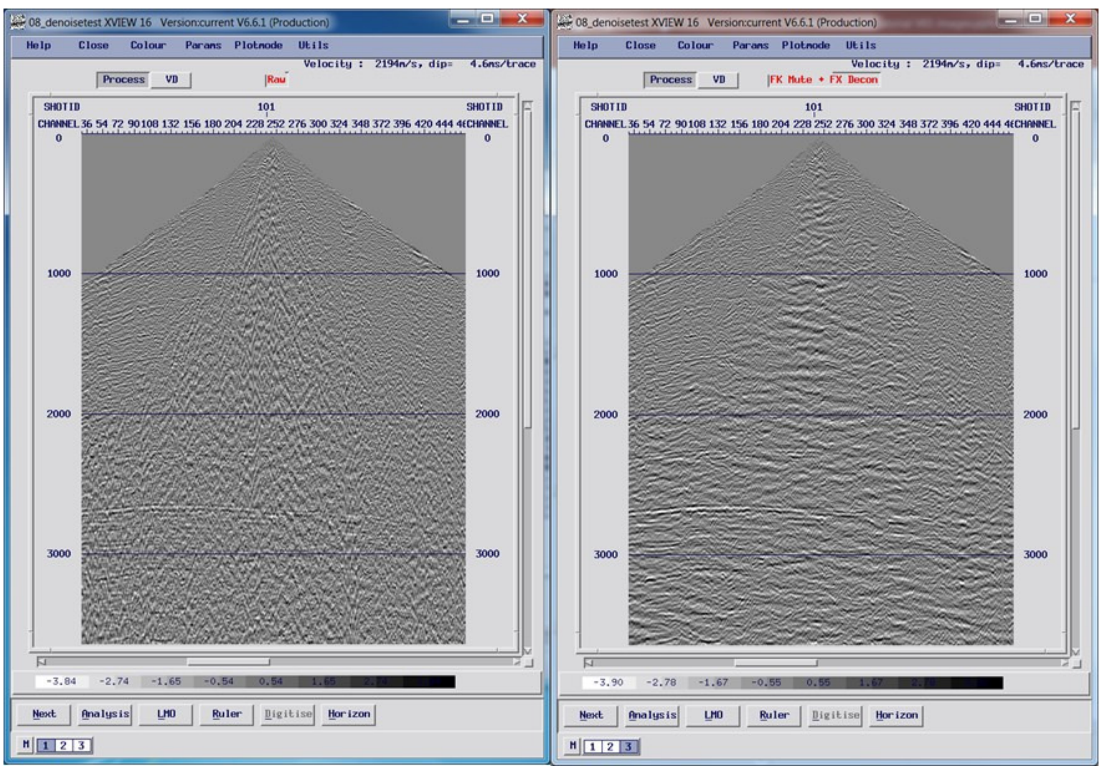
UNAGC removes the effects of the AGC using the saved scalars

PSEUDOMATH can also be used to merge the current data trace and a pseudotrace. In this case the output trace is made up of the unfiltered “copy” we stored earlier as far as the value of SPARE1, and the filtered trace below that.

PANELTEXT configures the labels for the XVIEW plot.

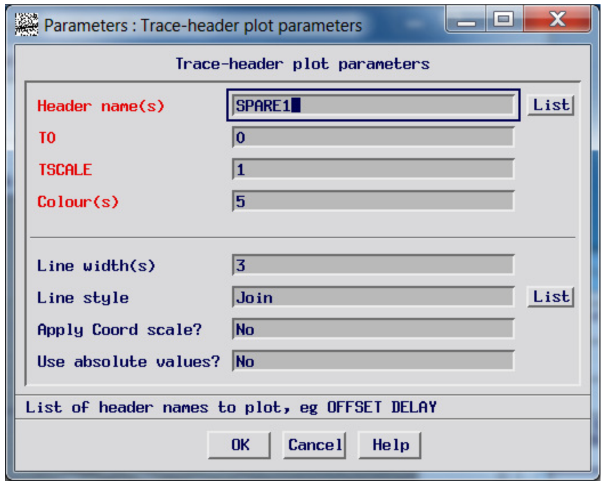
Run the job, and you will see the impact of applying the FK domain mute and FX Deconvolution random noise attenuation below the “noise cone” as a series of panels, along with the unfiltered shot records.

There’s still some amplitude variations, so to get a better view of the data, click on the Process button, fill AGC in for the processing sequence and click OK.



Results from the processing flow 07_denoisetest.job; on the left is the raw shot, and on the right is the shot with an FK Mute and FX deconvolution applied inside the noise cone.

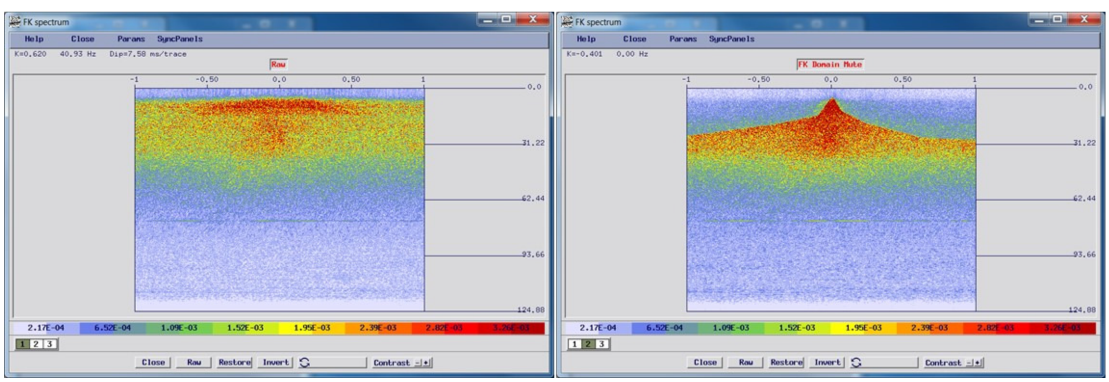
To see where the “noise cone” has been defined, you can overlay the SPARE1 trace header on the data – select Utils and ‘Overlay trace headers’; complete the form as follows:



Displaying a trace header overlaid on the data; headers can be scaled, or positioned relative to a given time value.

To examine the shape of the FK domain mute:

- click on Analysis
- Select FK Spectrum
- Click once on a single shot (on any panel)
- Click on AllPanels
- Click on SyncPanels



FK Analysis windows showing the raw data (left) and the data with the FK Domain mute applied in the noise cone (right)

The “circulate” button on the FK domain analysis window allows you to pick mutes in different ways, and test apply them. In this case the “above” mode was used, and the mute designed to preserve moderate to low dips and higher frequencies as the ground roll is steeply dipping and low frequent.

You can try different mutes out on the “raw” shot; once you have picked a mute (using the left mouse button to pick, and right mouse button to finish) use the “Invert” button to see the FK domain mute in action. Remember this is applied to the whole shot, not just the “cone”

You can modify the flow to try different parameters, or your own picked FK domain mute. You may also want to try the SEMBSMOOTH and other denoise modules – click on the ADD button in XSJE and review the options for 'Coherency filtering'.

The processing flow **08_denoisestack.job** has been configured to apply the same noise removal approach to all of the shots, as well as create an output stack dataset for comparison.

Run this flow, and then compare the stacked datasets we have so far by selecting them in SeisCat (remember to refresh the SeisCat display), right clicking and choosing "View with seisview"
02_brute.stack
04_restat.stack
08_denoise.stack

9.7 Deconvolution Before Stack

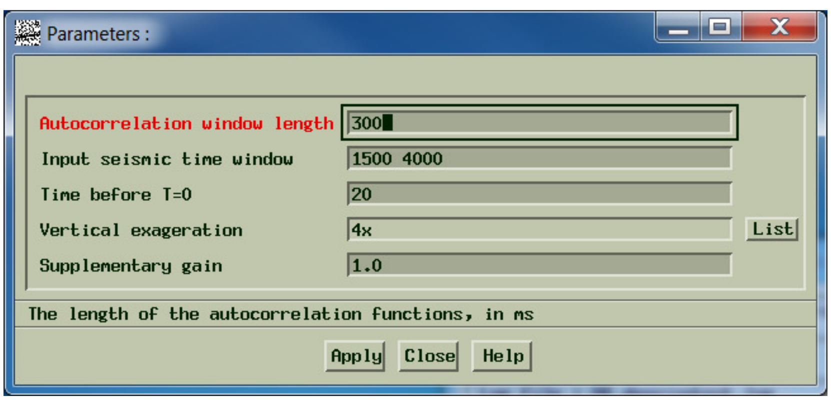
Deconvolution is another key processing step that potentially provides two useful outcomes.

The first is to compress any reverberant part of the wavelet, such as short-period multiples. The second is to change the wavelet shape, whitening it so that it has more frequencies and a sharper pulse.

We can start to explore what deconvolution will do by looking at the **08_denoise.shots** dataset. Select this in SeisCat and then right click to open the menu. Choose "View with seisview"

We can look at the reverberations in the data by appending an autocorrelation display.

- Click on Utils
- Select 'Add autocorrelation display'
- Click on Params
- Select the 'Autocorrelation functions' parameters
- Fill in the parameters as below:



Autocorrection parameters to look at reverberation in the shot records

You should now have an autocorrelation panel appended below each of the shot records.

SeisView lets you navigate forward and backward through the dataset using the <<, <, >, >> buttons.

The autocorrelation functions show very little reverberation, so most of the work we will be doing with deconvolution is to “whiten” the wavelet.

The processing flow **09_decon_tests.job** is designed to apply different types of deconvolution to selected shots from the cleaned seismic data.

There’s a few things to note in this processing flow:

SMUTE is being used to store the mute time as a trace header; this is then used as a reference for the start and end of the deconvolution design gates. You could also manually design these gates to vary with channel or offset, however this approach is much easier.

DECONW is using an “SDE file” to define the different operators and gaps. This file is a deconvolution widow file (dwf) and in this case is varying the deconvolution gap with the value of REPEAT. This means that each value of repeat will have a different deconvolution applied. Note that a gap of zero corresponds to a single sample or “spiking” deconvolution.

SCDECON is a “shot consistent deconvolution” – the autocorrelation functions are averaged across all of the traces within a shot record, rather than being calculated trace by trace.

In **XVIEW**, the panels are set up to be labelled based on the parameters from the DWF deconvolution file, automatically.

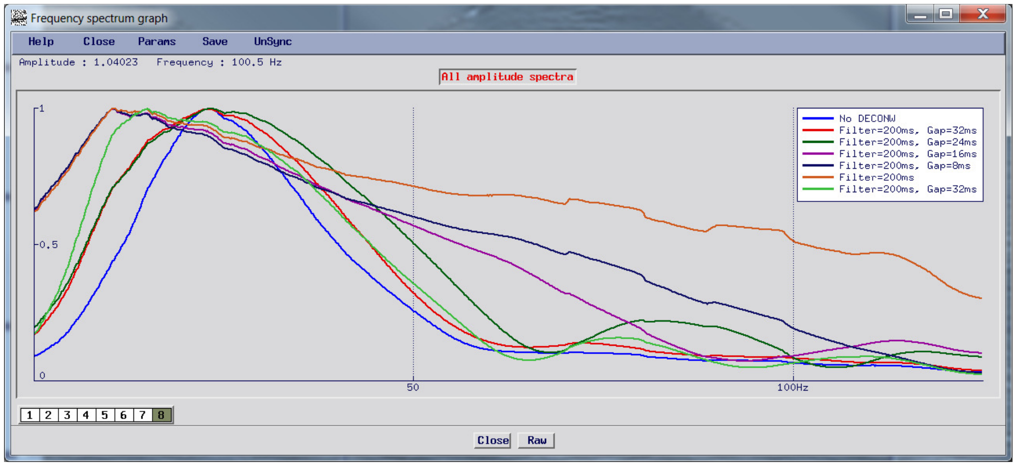
Run the processing flow, and look at the results.

As before, it is useful to apply an AGC by clicking on the process button, and entering AGC in the processing sequence.

To look at the results analytically, use the “Frequency spectrum graph” option that you can select after clicking on Analysis.

- Click once on a single shot record
- When the display opens, click on Params
- Set “Amplitude or Power” to Amplitude and “Mode” to Normalise
- Click OK
- Click on AllPanels and then SyncPanels
- Look at Panel #8

The last panel in this display (#8) overlays all of the frequency spectra from each of the REAPEAT panels so that we can look at the impact on the wavelet shape analytically.

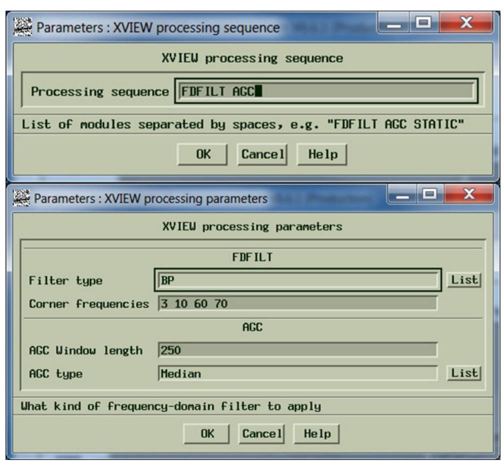


The frequency specta graphs from the deconvolution tests, all overlaid in the Analysis - Frequency spectrum graph window

In each case we can see how the deconvolution has "whitened" the spectrum, boosting frequencies outside of the original bandwidth. Where we have a gap that is too small, this has an "overwhitening" effect introducing high and low frequency ringing into the data.

Close the Analysis window, and click on the Process button with the middle mouse button.

Now add FDFILT before the AGC; this will let us apply a shaping filter after the deconvolution to remove the over-whitening and look at the waveform shapes



The 'Process' parameter forms, set to apply a filter, followed by an AGC to the displayed data.

You can specify the bandwidth of the filter as four points for the start and end of the low and high cuts.

When the processing sequence has been applied, you can examine the data and frequency content again.

The processing flow **10_deconstack.job** applies the 32ms gap, 200ms operator shot-consistent deconvolution to the data – with no shaping filter, and creates a set of shots for onward processing as well as a QC stack.

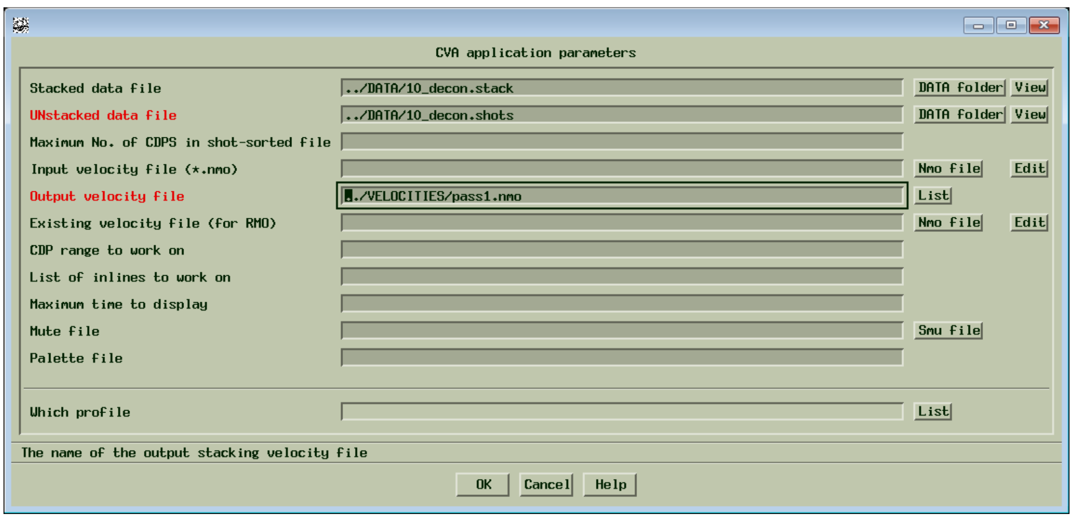
Review the stack along with the other stacks we have created so far, by selecting them in SeisCat, right clicking and opening with SeisView.

10 Velocity Analysis with CVA

Up until now we have created stacks using a simple brute velocity function. To improve the imaging further we need to make a more sophisticated velocity model.

CVA is the Claritas Velocity Analysis tool. With CVA there is no need to pre-calculate any data – you start with an unstacked seismic file (and optionally a stack) and can use that to develop – and QC – both the velocity field and corresponding NMO stretch mutes.

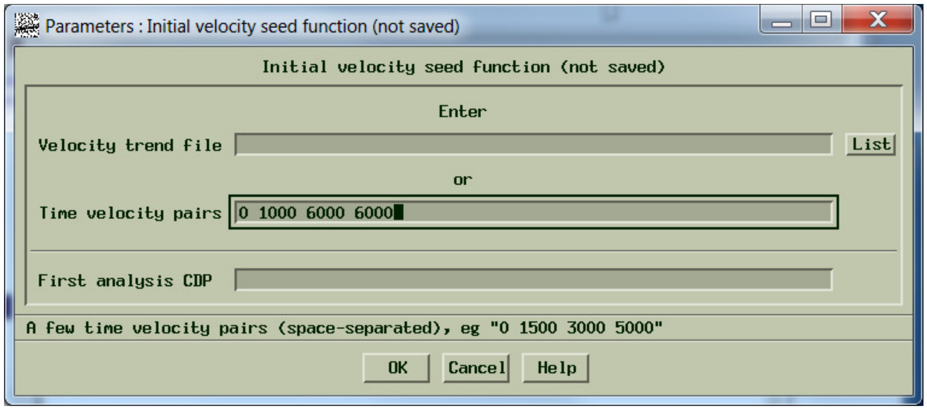
You start CVA from the launcher, under the Velocities tab.



The initial CVA form, selecting the stack and gathers created by the processing flow 10_deconstack.job

Note that the dataset we read can be in any order – it will be sorted to CDP automatically. We also don't need to have a stack to start – CVA can build a stack on the fly as needed. Call your output velocity file pass1.nmo and click on OK.

To get started we do need to provide some kind of velocity trend, this can be as simple as having two sets of time-velocity pairs:



Initial Time-Velocity pairs for CVA

10.1 CVA: Fundamentals

When we do velocity analysis we generally test-apply different moveout corrections to the data and view these in some way.

This might be in the form of a series of corrected gathers or a mini-stack, or it might be a semblance display where we show a measure of the trace-to-trace cross-correlation with different NMO corrections applied.

CVA has two main use modes that you can swap between at any time.

The first is based on **constant velocities**. This is useful where velocities might be highly variable, or where you don't have a good initial picture of the velocity profile.

When you use constant velocities, the semblance, stacks and gather displays are all calculated for a series of panels from minimum to a maximum velocity that doesn't vary with time, using a fixed increment between panels.

The second mode uses **variable velocities**. Here, the velocities used in the semblance, stack and gather displays are calculated around a central function.

Variable Velocity Approach	Constant Velocity Approach
You can efficiently pick the whole time range without losing accuracy	You have to choose between slow calculation time and the step between panels
If the velocities lie outside the "fan" you have set up, you might miss the trend	You can see the full range of velocities tested against all of the data
Best approach: Pick the whole trace, top to bottom	Best approach: Pick in time windows, layer stripping

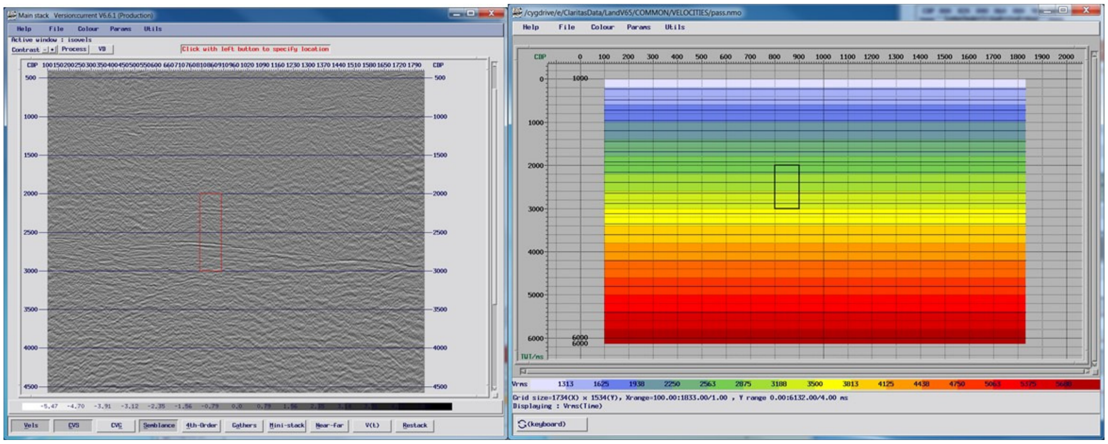
You can swap between "constant" or "variable" modes at any stage, however you do need to restart any analysis windows.

In CVA we can use "ministacks", "gathers" or "semblance" in any combination to pick velocities.

In this case, we'll start off using constant velocity stacks.

10.2 CVA: Getting Started

CVA has two main displays, the “Stack” window and the “Isovels” window.



The two main CVA displays – the stack (left) and Isovels (right) on start-up

At the moment there’s no velocity information – as we start to pick velocities they will appear on both the stack and isovels. The current isovels display is based on the trend we supplied.

Start by clicking on the stack; this will create a new velocity function where you click, and start the initial analysis.

The default mode is to create a semblance spectra, but for now, just click on close

Now we need to set up the constant velocity stacks.

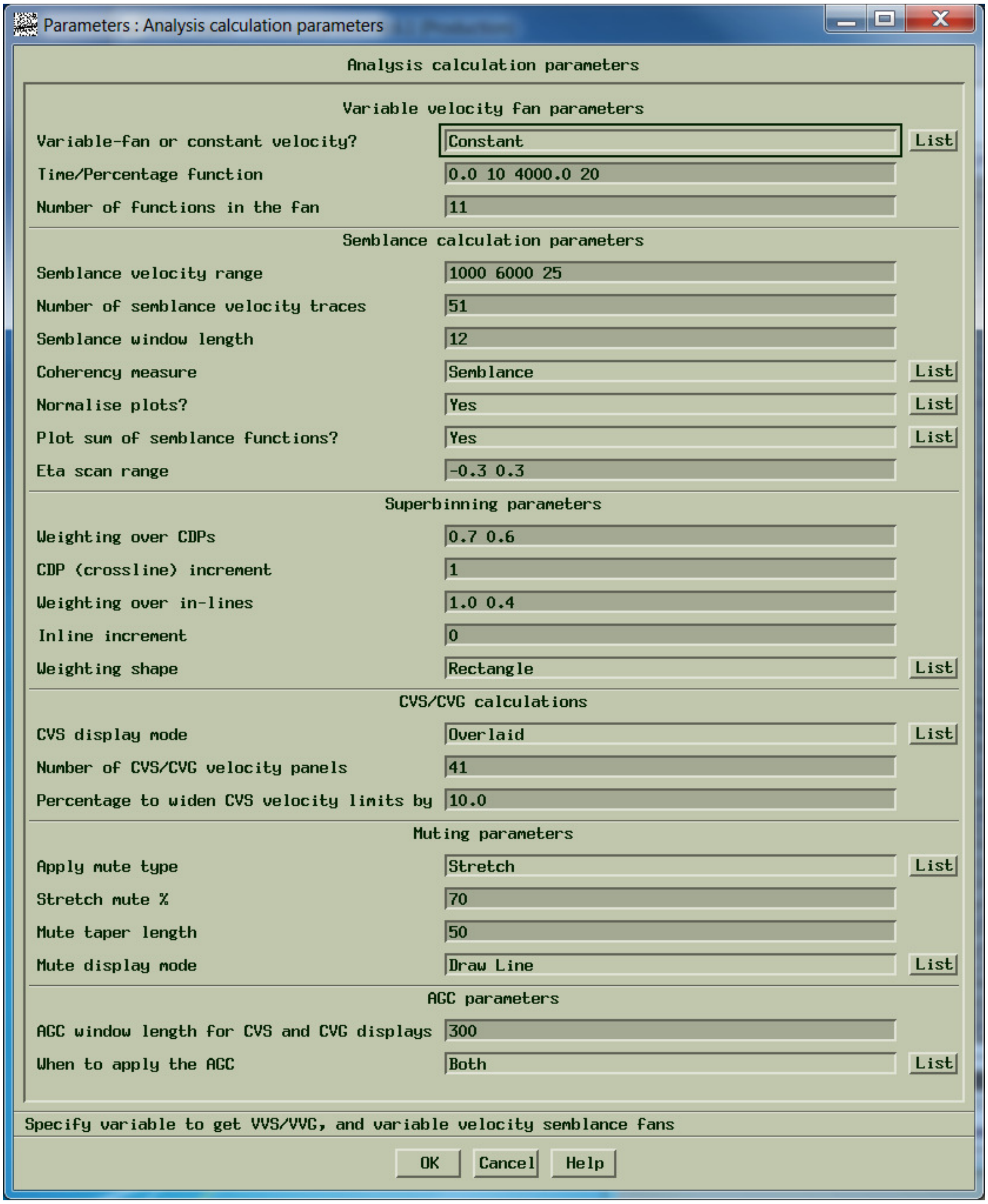
We’ll track across the data in a series of time windows, each with an overlap, so that we can follow events and pick them consistently – this is useful on land data, where semblance can be noisy or confusing.

In the main stack window click on Params.

Select **Analysis Calculation Parameters**.

This is a complex form, as it deals with the three main modes of picking – semblance, stacks and gathers – as well as how we want to deal with NMO stretch. For low quality data we also have the option to “superbin” gathers to improve semblance.

This is also where we choose between constant and variable velocity model. Set this form up as below (it should default to this)



The "Analysis Calculation Parameters" form, used to select between constant and variable models, as well as the parameters for all of the analysis types

Next, we need to set up the analysis positioning.

Again, click on Params.

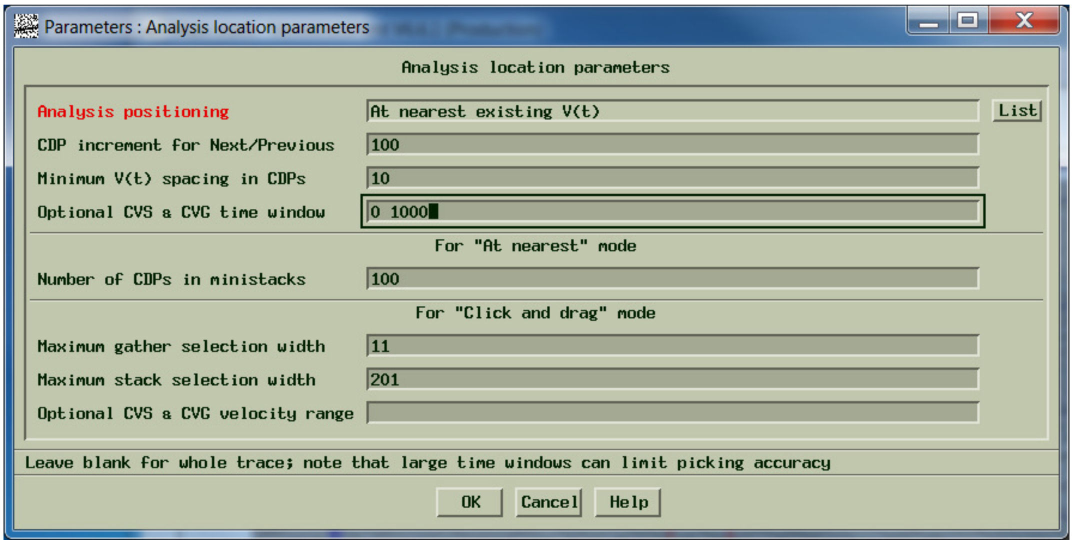
Click on **Analysis Location Parameters**.

The first form controlled HOW we were going to do velocity analysis. This form controls WHERE we will do it.

CVA is extremely flexible – you can perform velocity analysis exactly where it is needed, using drag-and-drop windows on the stack, if you want to. However, initially we want to pick at a constant fixed spatial increment.

Once we have started picking, we can then step forward or backwards across the section. This will either jump to the next function that exists in that direction, or it will jump by a fixed increment, whichever is smaller.

We're going to be layer stripping, so initially we want to set the positioning form as follows:



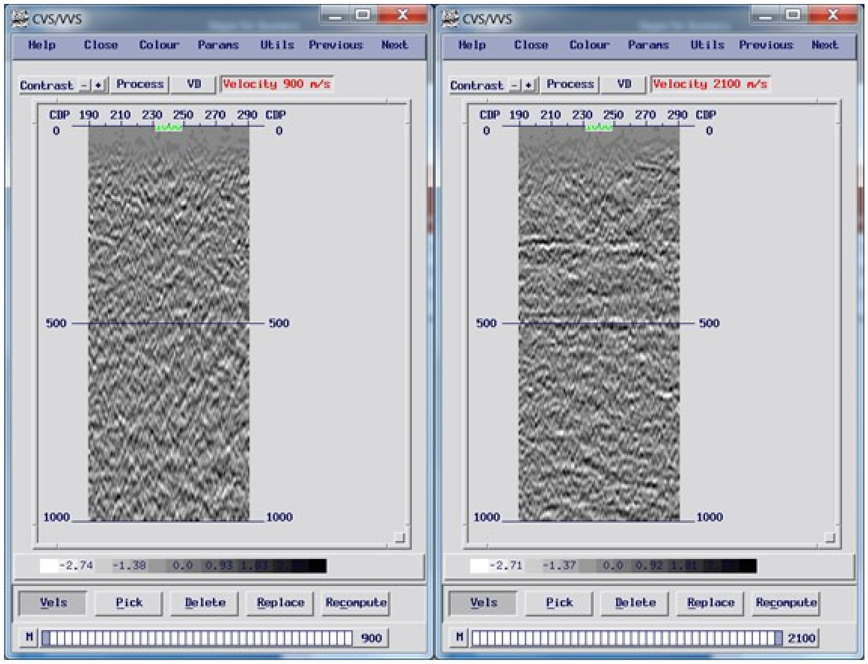
The initial set up for the velocity analysis positioning form; note the "Optional CVS & CVG time window" and "Number of CDPs in ministacks" parameters.

10.3 Starting Picking

Click on the CVS button, and the click on the stack close to CDP 200.

This will open a display window with a series of panels – 41 in total, which we set up on the calculation form. In each panel a different constant velocity NMO has been applied to 100 cdps, and the data stacked.

The initial range for the velocities is 900m/s to 2100m/s – however its only on the last few panels that events starts to appear:



The 900m/s and 2100m/s constant velocity stack panels; we only start to see events "stacking in" on the last few panels in the CVS display

To fix this, use the Recompute button, and enter 1500 2500 as the velocity range.

Now when you scroll through panels (using the mouse and holding down the left button) you can see the event at ~300ms appear, then vanish. The stack is strongest around 2075m/s.

Click on the Pick button, and pick this event on the 2075m/s panel.

When you do this, the value will appear on the main stack window, the ministack (if the 'Vels' button in selected), and the Isovel plot updates.

In practice it is much easier to use the "hot keys" to control the picking, deletion or replacing an existing pick with a new value.

The underlined letter (P, D, R) on the buttons is the keyboard hot key; CVA is designed in this mode to be used two handed, with one hand on the keyboard, the other on the mouse.

There's also an event at 500m/s which stacks in at ~2150m/s and around 750ms you can see an event at 2300m/s.

Pick these, then hit "Next" on the toolbar menu to step along the section.

When you press Next, CVA automatically interpolates a new function – in this case a copy of what we just picked. If the structure is similar, we will only need to **Replace** velocities – which can also include adjustment in TWT.

You may need to **Recompute** the range again so that it is from 2000m/s to 3000m/s.

Step across the data, picking the velocities in the first 1000ms; use the Isovels as a guide.

Tips:

- Animate the stacks forwards and backwards to see how events “come in” and “fade out” with the velocity, to help you to pick
- Make sure you don’t pick too close together; 80ms TWT is probably a minimum
- If you click on the CVG button, you will get a synchronised constant velocity gather window with corresponding NMO corrected CDP gathers
- If you double click on a velocity location on the stack, you will jump to that location
- The reflectors may still not be very hyperbolic as a result of near surface velocity variations; we’ll work on that later with residual statics

When you have reached the end of the line, go back to the “Analysis Location Parameters” and adjust the time window so that it is from 750ms to 1750ms.

Close the current CVS (and CVG) windows, then click on CVS and the first location you picked; keep in mind you may need to reset the velocity range as you pick.

Carry on “layer stripping” in this way, with a small overlap between the time windows. You can increase the size of the time window as you get deeper, as velocities become less sensitive with depth.

Below about 4500ms TWT there’s very little moveout discrimination and it’s hard to pick.

10.4 Checking Your Picks

When you have a velocity field you can click on the Restack button on the main stack window. This will create a new stack panel with the updated velocities applied.

You can toggle the display between the two stacks – it may help to turn off the velocity overlay with the Vels button.

If the stack has degraded after you picked velocities you may need to go back to that location and repick.

You can also check the picks on the Isovels display. Clicking on the circulate button opens up options to add, move or delete picks (and functions) or to replace velocities with an entered value.

10.5 Refining Your Picks

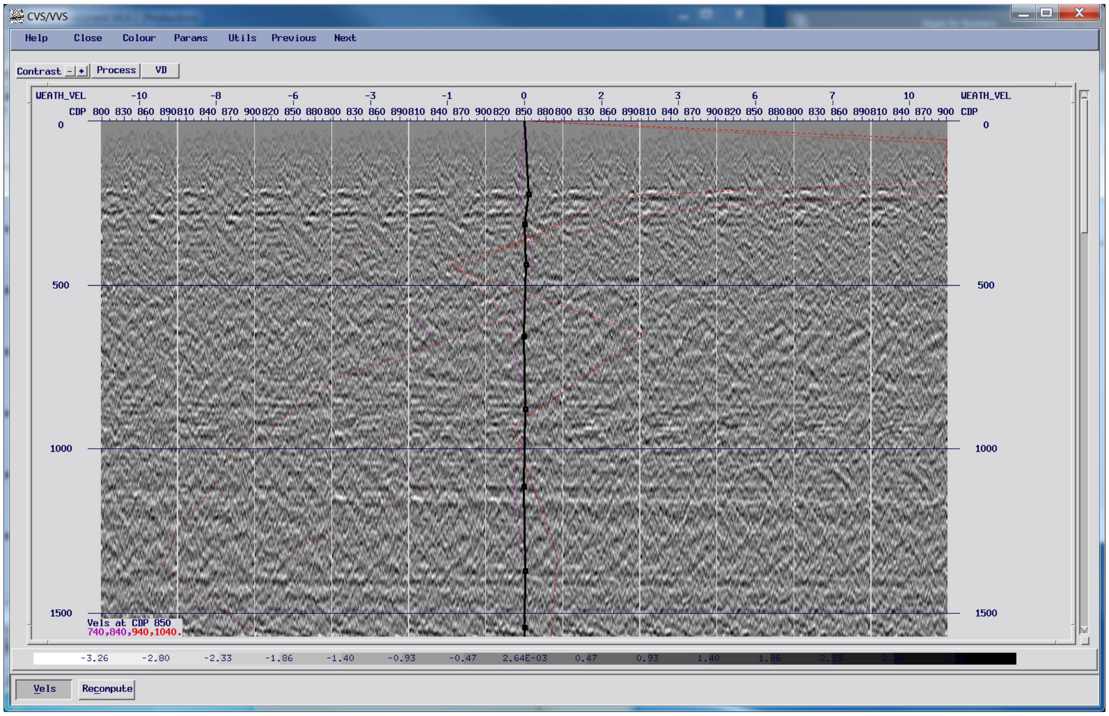
To refine the picks, lets change mode and use some other techniques.

Select “Variable” in place of “Constant” velocity in the Analysis Calculation Parameters form, and select CVS display mode as Side-by-side not Overlaid.

On the Analysis Location Parameters form delete the “Optional CVS & CVG time window” parameters.

Now when you start VVS, you will see a set of variable velocity stacks, displayed side-by side.

The velocity at the previous and next locations are also shows as dotted purple and red lines.



The VVS Variable Velocity Stack display in side-by-side mode

The current velocity function is shown as a back line; you can pick this in the same way we picked the refraction mute, with the left mouse button. You can drag velocity control points over to the strongest stack display response, or delete them and repick.

The "Recompute" button will re-centre the velocity function on the central stack.

When you are happy with the velocities, go to "Save and Exit" under the "File" menu on the main stack.

Note that there's a lot more functionality inside CVA – including picking 4th order velocities and NMO stretch mutes; the online manual has full details of the different modes and functions you can use.

10.6 Velocity QC Stack

The processing flow **11_velsstack.job** creates a stack with the new velocities - you'll need to modify the flow to read in the velocity file you created. In this job, note how SEISREAD is sorting from the original file order of SHOTID and CHANNEL to CDP and CDPTRACE.

Run the job, refresh the SeisCat list and review all of the stacks created so far.

11 Residual Statics

11.1 Overview of Residual Statics

While we have tried to correct for the varying near surface geology and topography through elevation and refraction statics, we have seen that many of the reflections on the pre-stack gathers are not clear hyperbola following the Normal Moveout equation.

Residual statics are designed to help the data conform with the NMO equation by creating shot and receiver surface consistent static shifts that remove the nonlinearities from NMO corrected gathers.

The residual statics process starts off with a “pilot” – usually a smoother version of the stack. We supply NMO corrected data to the algorithm and each trace is cross-correlated with the stack (at the relevant CDP location) and a shift generated; these shifts are then decomposed into surface consistent shot- and receiver components just as we did with the residual error from the refraction statics.

The process is iterative, with only small shifts being applied, and is designed in this case to optimise the stack power within a given time window. Periodically the inversion will create a new “pilot stack” automatically, so that the final statics can continue to converge.

Careful QC is needed to avoid “cycle skipping” – where different parts of the waveform are forced to align – or removing geological features like faults.

11.2 Running Residual Statics

The residual statics processing flow **12_resids1_run.job** is a little different to those we have used before. The first and only module in the job flow is the SPSTAT2 module, this is designed to be both and input/output and processor module.

The SPSTAT2 module creates the pilot stack on the fly and can apply the NMO correction within the module so there is no need for a job flow to prepare the data prior to residual statics, the user can also apply optional noise attenuation processing to the model if the input data is noisy and can define and apply an AGC if required to the input data only, model data only, or both.

The actual residual statics algorithm used in this flow is SPSTAT2, a stack-power maximisation approach. The module is currently set up to use a constant fixed time window from 550ms to 3200ms TWT, however you can also define a spatially varying window if desired.

When you run the processing flow, it will produce a series of statics files; with 8 iterations there will be 16 files in total.

The first set of files is the residual statics produced by each iteration – these will be in the STATICS folder and called:
resids1_001_res.shf
resids1_002_res.shf
and so on.

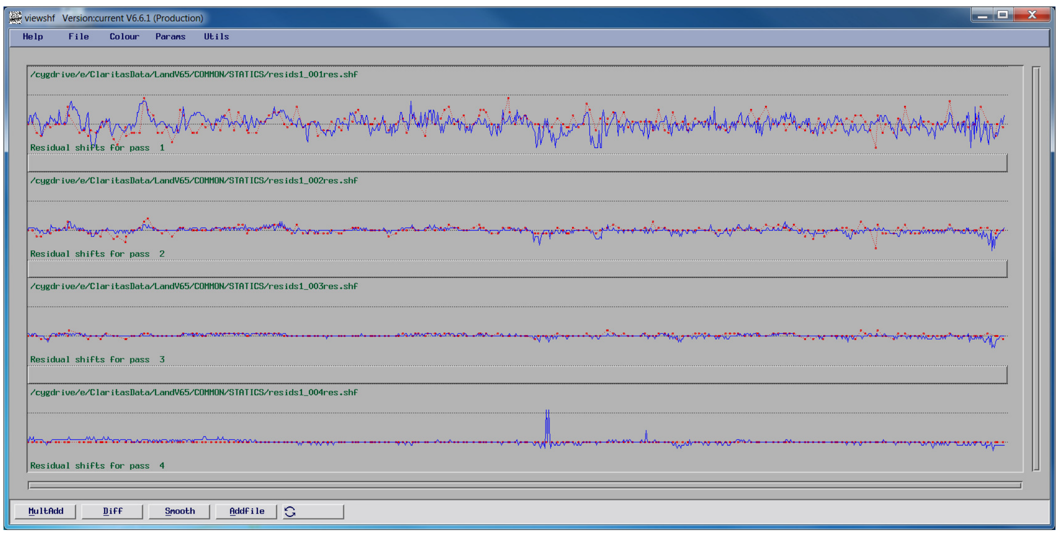
The second set are the cumulative shifts at the end of each iteration, which are called

resids1_001_cum.shf
resids1_002_cum.shf
and so on.

11.3 Checking Residual Statics

We can review these outputs using the ViewShf application, this is automatically launched by the **12_resids_run1.job** as the VIEW_SHIFTS parameter in the SPSTAT2 module is set to Yes. The files displayed are for the iterations defined by the SAVE_STACK parameter, the application allows you to see the shot and receiver statics displayed.

To start with, look at the "res" residuals files.



The first four iterations of residual statics; blue line is receivers and red line is the shots. Most of the work is done by the first and second iterations. The spikes on the forth iteration are not a "cycle skip" issue, but linked to the maximum shift that can be applied in any one iteration.

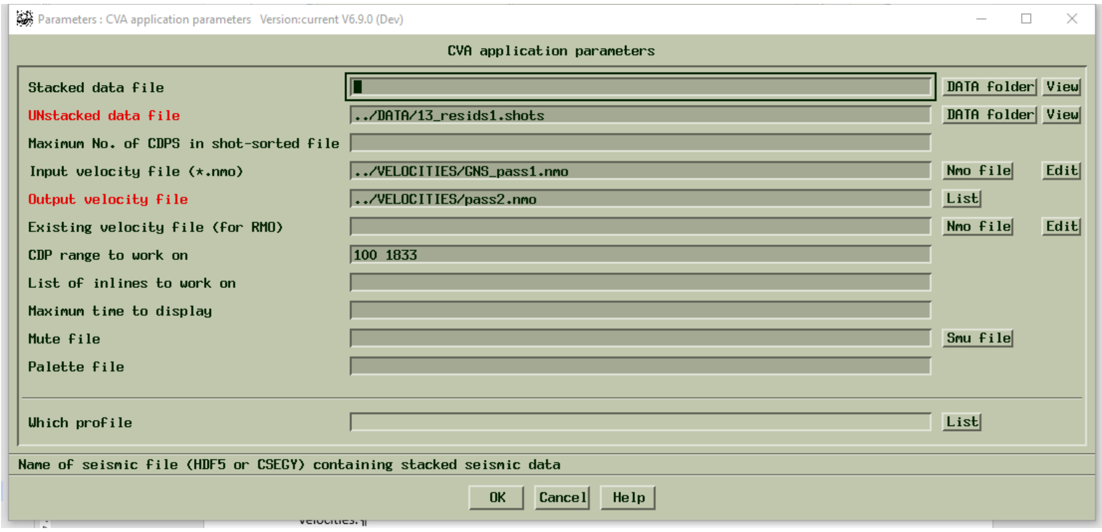
We expect to see the longer spatial period issues addressed in the first few iterations, with smaller changes appearing as the solution converges. In the case above there is a big shift in the fourth iteration, however it is spatially localised and not likely to be a result of cycle skipping.

The SPSTAT2 application will also launch two SeisView sessions, one will display a stack for the iterations defined in the SAVE_STACK parameter so you can evaluate the effectiveness of the Residual statics solutions produced, the second session displays the Model used as input to each iteration of the residual statics process. Using the panel buttons you can easily see the changes as the inversion progresses.

11.4 Second Pass Velocities

After running residual statics, we may be able to make further improvements to the velocity field.

Run the processing flow **13_resids1_apply.job** to apply the statics, and then start CVA reading in these data and the first pass velocities as a guide; call the output **pass2.nmo**



The CVA form configured to pick the second pass velocities after residual statics has run.

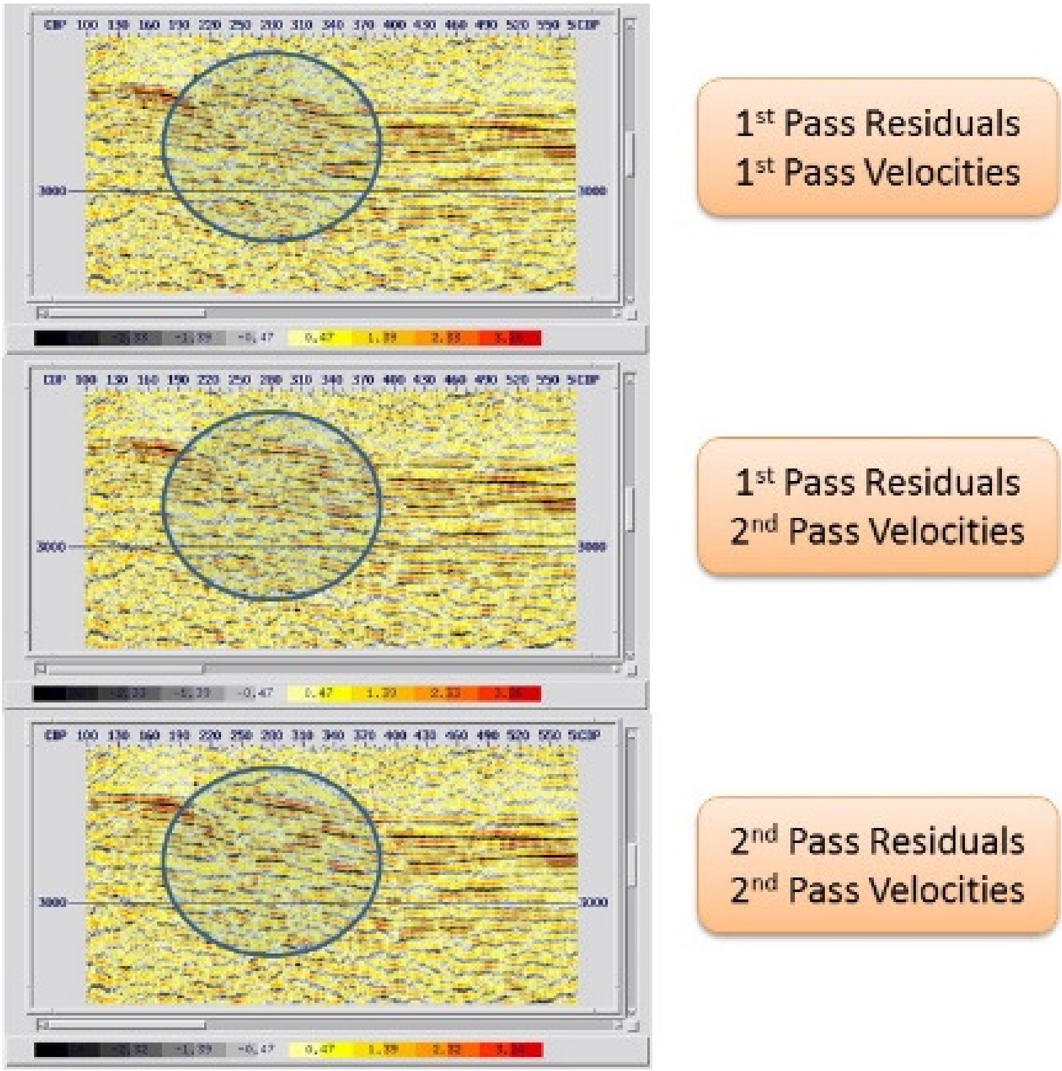
You'll be prompted for mute and AGC parameters – just click on OK.

11.5 A Second Pass of Residual Statics?

It is not unusual to apply multiple passes of residual statics, with an updated velocity analysis between each one. You can do this if you choose, building your own flows from the examples provided. The steps will be

- apply the final residual statics to the data and output shots
- read these shots into CVA, using the previous velocity field as a guide
- pick velocities
- run the residual statics inversion
- check the results

In this case a second pass of residual statics may well improve things, along with another round of velocities.



Progressive improvement with 1st pass velocities and residual statics (top), repicking velocities after the 1st pass residual statics (middle) and running a second pass of residual statics (bottom)

12 Imaging

Seismic surveys record the scattered and diffracted wavefield. If the sub-surface structure is simple and relatively low dip this is a reasonable representation of the geology, however in most cases prior to interpreting the data we need to correct for dip, and “collapse” any diffractions.

The general process for doing this is termed “migration”, where we seek to migrate the seismic energy back to something that better represents the sub-surface position it originated from.

Broadly speaking, we can apply migration in the time domain or the depth domain.

Time migration is quicker and utilises a less sophisticated velocity model, however it doesn't correct for the effects of refraction at interfaces between layers, or within rocks that have strong horizontal or vertical seismic velocity gradients.

We can also apply migration to pre-stack data or post stack. In general, pre-stack applications require an even distribution of offsets (as we tend to migrate by offset plane), good signal to noise ratio and a good velocity model.

In this case we have some complexity; it's hard to constrain the velocity model, and we have areas with low signal to noise.

For that reason, in this tutorial we'll use Dip Moveout or DMO. DMO is a partial pre-stack time migration with a weaker dependence on the velocity model, which also tends to produce fewer artefacts as the signal: noise ratio is weak.

12.1 Applying DMO

The main practical difference between DMO and other migrations is that the operator shape is not directly affected by the velocity; instead we apply DMO after first applying NMO, and then reverse the NMO correction.

The implementation of DMO we use also requires the data to be sorted to common offset planes prior to its application; we can do this with SEISREAD when working with HDF5 files, however if you are using the older CSEGY format (created by DISCWRITE) you can use the DSORT_OFF processing module.

Sorting to offset planes is a "binned sort" – we create a series of "offset bin centres" and allocate traces to the bins based on the bin centre and a half-width.

While the output from DMO will still be in offset-plane order, this can be read directly into CVA for the next velocity analysis phase.

An additional velocity analysis may be needed because the impact of dip on the seismic image (and velocity) is addressed, at least in part, by DMO. As a multi-trace migration process, DMO can also help to boost coherency in noisy data.

The processing flow **14_DMO.job** applies DMO to the data; note that this is reading in the output from the **13_resids1_apply.job** processing flow and using the second pass velocities.

Key things to note:

- the SEISREAD module is set up to sort to offset planes on input
- the NMO applied (and removed) is GNS_pass2.nmo
- the DMO module includes offset binning as well
- SEISWRITE will output the data in common offset planes
- This job may take some time to run, perhaps >1-2 hours.

If you ran further iterations, you will need to modify the flow accordingly.

The velocity file **GNS_pass3.nmo** and the statics file **GNS_resids2.shf** are the "second pass" residual static and a third pass of velocities, which you could modify the processing flow to use.

12.2 Velocities after DMO

As before we can use CVA both to create a new stacked output of the data with DMO applied, and repick velocities and the NMO stretch mute.

In this case, you should select the dataset **14_dmo.offsets** and use appropriate input and output velocity files. An example of a velocity field picked after DMO is saved in **GNS_pass4.nmo**.

You can use the restack button to compare the results of your velocity analysis.

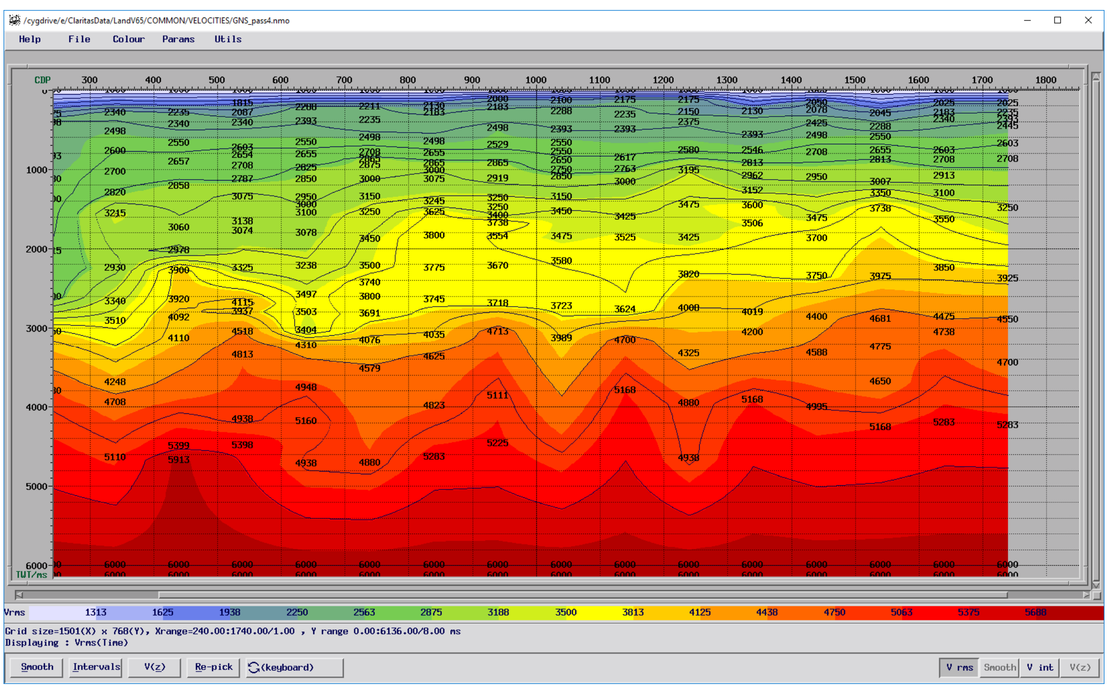
12.3 Post Stack Migration

We're going to run a post-stack finite difference time migration to complete the imaging phase. This means that we will need to stack the data, as well as apply some scaling to reduce edge effects.

The FDMIG module uses interval velocities, and we can apply a scale to those velocities to help us to optimise the image.

12.4 Building the Velocity Model

On the Velocities tab in the launcher click on the isovels button and load up the post-DMO velocities that you have picked. The Isovels application allows us to manipulate and edit a velocity field in various ways.



The post-DMO velocity field in ISOVELS; this is the GNS_pass4.nmo field

If you have picked some velocity inversions – and the combination of velocities and statics may lead to this – then you will need to address these first.

- Select "Delete Bad Vint" under Utils
- Specify a range of 1000 6000
- The process will tell you how many picks were deleted and iterate automatically – click OK to run again and repeat the Vint rejection

When that has been done (and there are no more picks to be deleted) we can smooth the velocity field.

- Click on the Smooth button
- Select 'Filter' for the "Smoothing methods to apply"
- Select Both for the "Smooth by 1/v in" option
- Select a Filter length of 61 (Method 3 : Low-pass filter)
- Click 'OK'

And finally we can convert to interval velocities and save the file.

- Click on Intervals button
- Set "Which times to use" to 'Grid'
- Specify the regular time grid spacing as 25 (ms)
- Click OK
- Click on Repick
- Use 'Regular grid' for CDP positions and time positions
- Use 25 for both the CDP and time grid spacing
- Click on the List button for output file name, and call the file mig_model.nmo

If you are reviewing the gridded model in Isovels, it is useful to turn off the plotted velocity picks; this is an option under 'Params' and 'Plot parameters'.

12.5 Creating and Scaling the Stack

Migration processes rely on an operator that creates a new image through constructive and destructive interference. This can lead to artefacts if there is a lot of variation in the trace amplitudes, as well as at the edges (sides and bottom) of the seismic section.

The processing flow **15_scaled_stack.job** demonstrates how we can use scaling to mitigate these two effects.

The trace-to-trace amplitude variation is addressed using the "BALANCE" module; this is applying a "rolling gate balance" with a series of 1000ms windows, overlapping by 50%. This evens out the amplitudes in a gentler manner than an AGC, which can also have a negative impact on the migration operator.

The balance gates are in a separate spreadsheet file, **15_rgb.btw**, and is applied pre-stack.

To control edge effects a time and space varying scale is used. The file **15_mig.scl** controls this scalar, which ramps down the amplitudes at the edges and end of the dataset.

The processing flow creates and migrates a stack, with and without these types of scaling.

The final balance ensures that the three panels can be easily compared.

12.6 Using JCS to Test Migration Scalars

The size of the migration operator in a post-stack migration depends on the velocity field.

If the velocity is too high we will over migrate the data, leaving “smiles.” If the velocity is too low, then diffractions will not collapse correctly, and key features like faults will be unfocussed.

One way we can run tests like this is to use the JCS job control system. With JCS you can create a master processing flow which includes named variables inside a pair of angle brackets like this: <***variable_name***>

We can then duplicate the processing flow and substitute the variables for values – which can be numerical or text – creating a suite of processing flows. These are all given a unique name, and will automatically execute one after the other.

JCS can also be used for applying the same processing sequence to many lines, by varying the input and output file names and any line specific parameters like CDP.

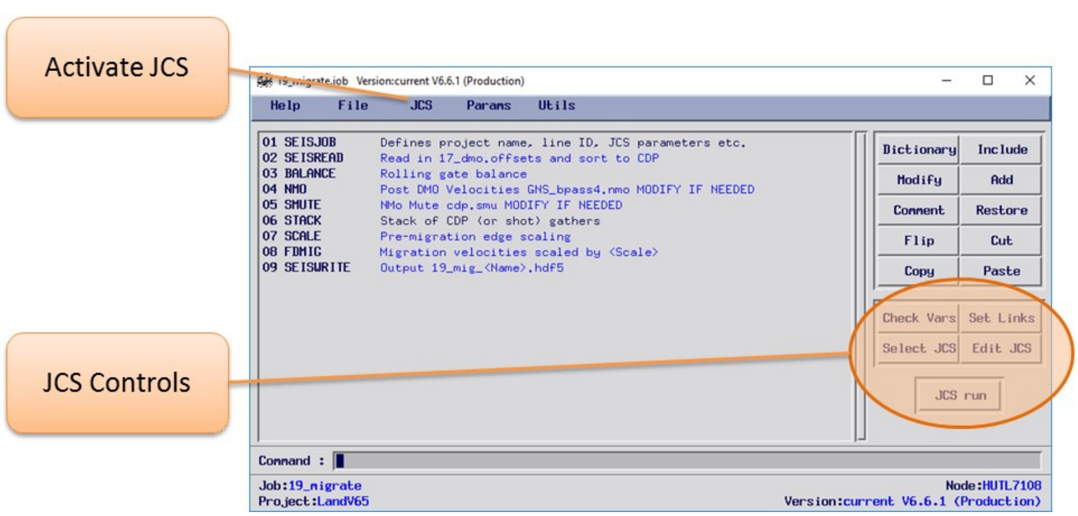
Open the processing flow **16_migrate.job**.

The FDMIG and SEISWRITE are both configured with variables inside angle backets.

In FDMIG the migration scalar (SCALAR) would usually have a numerical value, however in this case it has **<scale>**. In SEISWRITE the name of the output file is given as **16_mig<Name>.hdf5**

If you try to run this job it won’t work, because the text **<Scale>** appears where the flow builder is expecting a numerical value.

To activate “JCS mode” click on the JCS tool bar menu option, this activates the JCS controls.



Processing flow 16_migrate.job with JCS mode activated; five extra buttons are added to the flow editor.

In this case JCS has already been set up to run; you can explore how this works by:

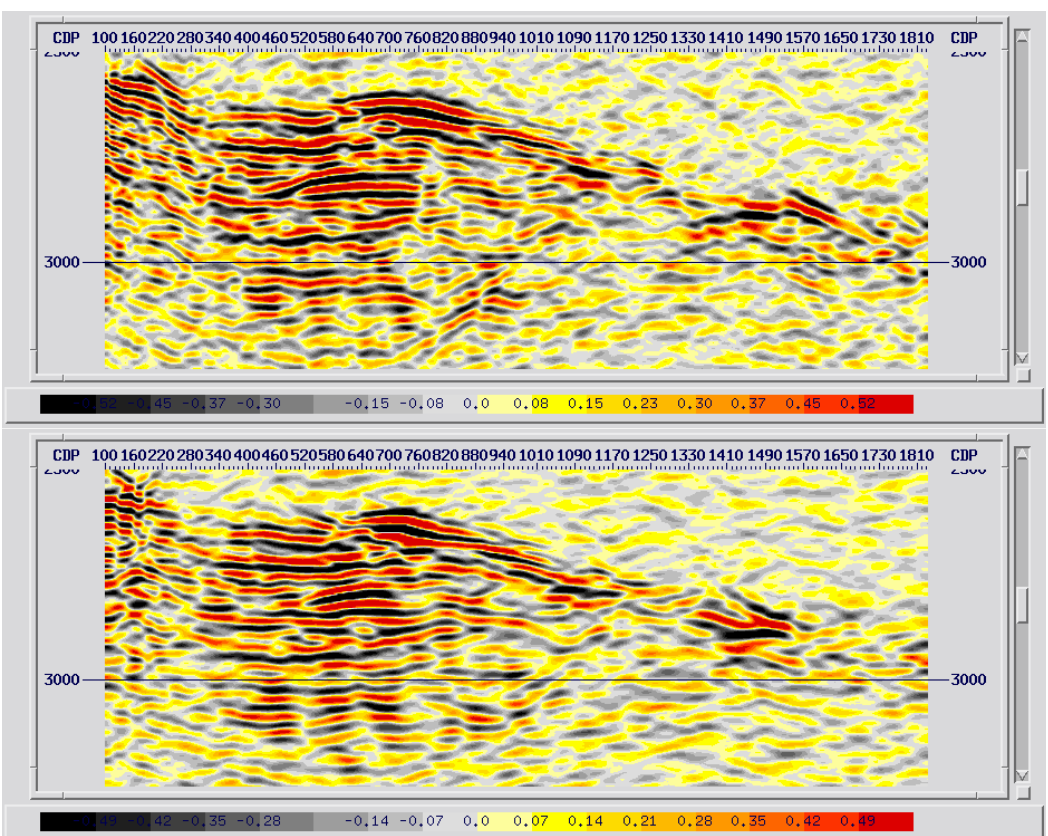
- **Click on Edit JCS**; this will open the table of variables that we will substitute into the flow. While **SEQnum** and **Name** are mandatory column names, you can modify the others by clicking on **Commands** and then selecting **Rename columns**
- **Click on Check Vars**; this scans the processing flow looking for variables in angle brackets <>; this will tell you that all the variables are identified.
- **Look in SEISJOB**; this special module contains the link to the JCS spreadsheet file, as well as the names of the variables that have been found, in this case <Name> and <scale>
- **Click on Set Links**; JCS will automatically match the variable in the flow against any column in the JCS spreadsheet with the same name; in this case we have a slight mismatch between **<scale>** and **Scale** (the capital letter) so we're asked to make the link. The links are also saved in **SEISJOB**
- **Click on Select JCS**; this is to identify which of the parameters you want to run. Click on "SelectAll" and "OK"
- **Click on JCS run**, then OK

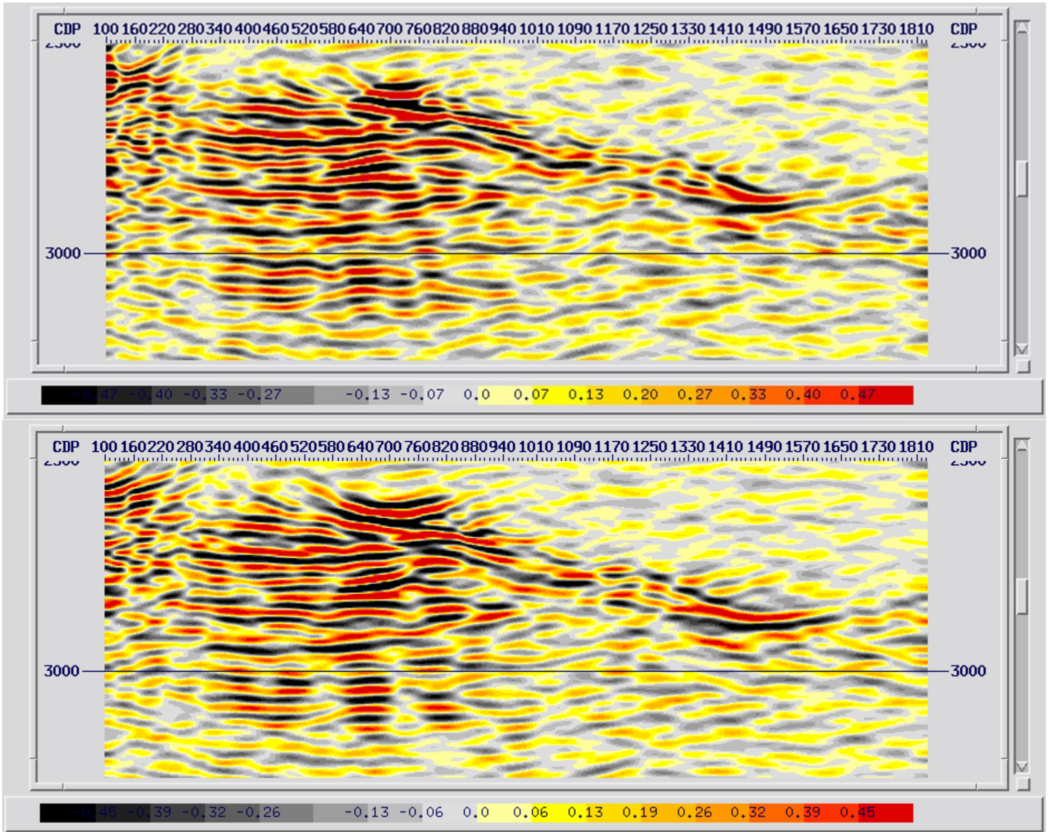
JCS will now create 16 separate processing flows (each numbered from 1 to 16 based on the SEQnum column in the spreadsheet. These flows will then run, one after the other.

12.7 Seeing the JCS Results

If you open SeisCat and refresh the file listing there will be 16 new files, one created by each processing flow.

You can select all of these files and view them using SeisView.





Migration images with the velocities scaled by 50% (top), 100%, 150% and 200% (bottom). Note the overmigrated nature of the fault in the bottom image, around CDP 780.

13 Finalisation

After migration we usually apply some additional processing stages to further improve the signal:noise ratio. Modern interpretation workstations can also have this functionality, so these stages might not always be required as part of the processing phase.

13.1 Filtering

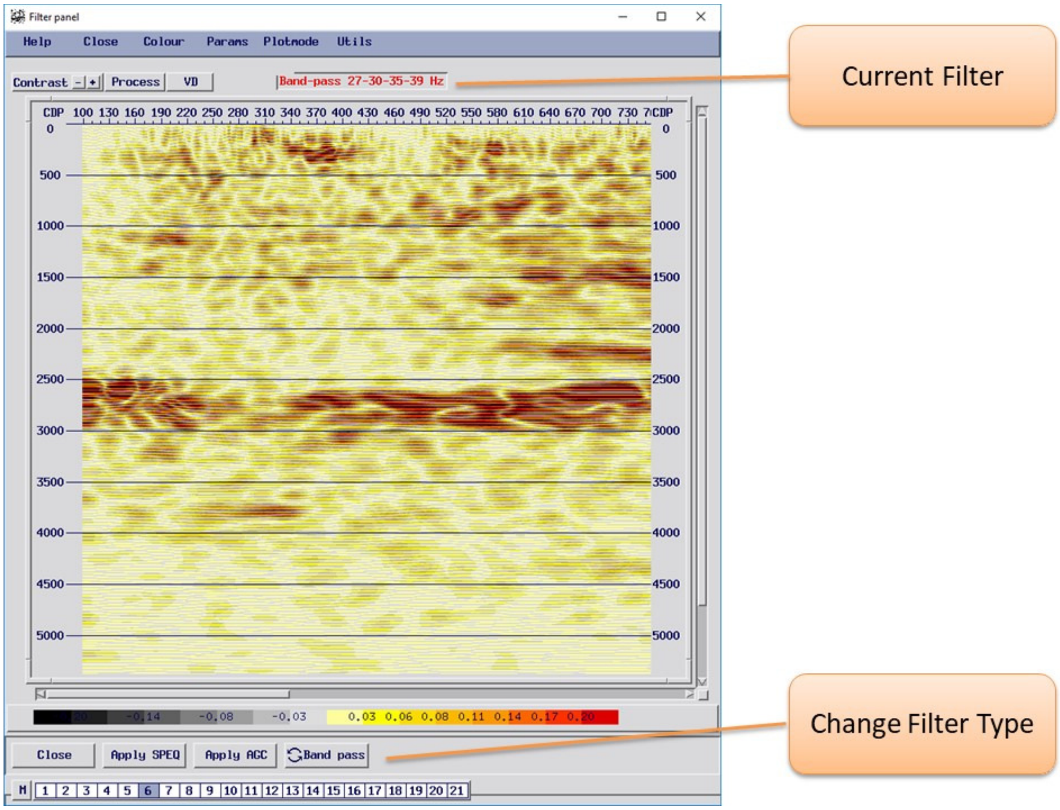
High seismic frequencies tend to get absorbed and scattered in the near surface. As we look deeper in the section it becomes more likely that the higher frequencies are noise, not signal.

We can review this interactively in GLOBE Claritas.

Pick your preferred migration in SeisCat, and then right click to open and view it with SeisView.

Next, click on the Utils menu and select "Create a filter panel"

This will open a new window with a series of (initially) 21 panels, each filtered with a different bandpass filter.



The filter panel display. You can select band-pass, high-cut or low-cut filter types.

While the initial display is a bandpass filter, it can also be useful to review low-cut and high-cut filter panels to help determine the noise characteristics of the data. The filter type can be changed by right clicking on the circulate button (with the two arced arrows) and selecting the desired option.

In this case there is very little data below the 69Hz-74Hz high cut band.

13.2 SEGY Output

In general, the last thing we do in processing is to create a SEGY format dataset. This is the Society of Exploration Geophysicists "exchange" format, which can be universally read by all seismic interpretation software.

Open the processing flow **17_export_segy.job**.

Use SEISREAD to read in your preferred migration from 16_migrate.job.

The SHOTPOINT module can be used to add the shot-peg values into the trace headers. Before stack, SHOT and SHOT_PEG refer to the shot ID and position for each trace, but after stack, those values are meaningless, and a new shot-peg value is required which gives the equivalent shot-peg value at each CDP position. SHOTPOINT does this using the information already in the geometry database.

The DATUM_SRD module applies the second part of the floating datum correction, as started by the DATUM_FLT module – see section 5.4, correcting from the floating datum to the seismic reference datum.

The ADDLLD module can then be used to add the Lat/Lon or Easting/Northing for each shotpoint into the trace headers. ADDLLD requires a text-file to be created, with a list of shotpoints and their coordinates.

Use RHEADER (or RHEADER2) to fill in the 3200-byte EBCDIC headers, this module can take a specified text file and store this in the 3200-byte SEG_Y character reel header. This header contains information such as location, date, details of acquisition, and applied processing.

In the WRITESEGY module enter the name for your final SEG_Y output, and specify big-endian and IBM format output (ie, SEG-Y standard).

The client may also require certain trace headers to be set in certain byte positions, this can be done by using the DEFNAME, INLOC, and INLONG parameters in WRITESEGY. In 17_export_segy.job, these parameters are used to write the source point (SHOT_PEG) and CMP coordinates (CDP_X and CDP_Y) in bytes 9:12, 181:184, and 185:188.

You can check that the new header entries are correct by selecting the newly created SEG-Y file in SeisCat and opening in SV. Then the ‘Utils’ – ‘Overlay trace headers’ option can be used to verify that the shotpoint and coordinate values look correct.

The SEGY analyser tool can also be used to check that the SEGY has been correctly written, this is started by using the ‘Analyser’ button on the ‘Seismic Data’ tab of the launcher. You can review the binary header, or trace headers, as well as displaying sample amplitudes as a histogram or an XVIEW display of the data. Options allow “non-standard” SEGY data to be analysed, and either rewritten or a flow created that will read them in.