AGS Aarhus GeoSoftware

Workflow for import, processing, inversion and presentation of DC/IP data in Aarhus Workbench

This guide presents the common workflow options from workspace creation to final presentation of data for full decay DC/IP data in Aarhus workbench. Aarhus Workbench contains a wealth of options and possibilities and not all of them are presented in this manual, but all the basic options and many of the advanced options related to working with ERT data are presented.

The flowchart is color-coded, the deep blue boxes are the core task that will always be carried out, the ordering is to start from the top and work downwards. The orange boxes are optional extras that can be carried out if the relevant data are available. The light blue boxes are quality control and presentation tasks of which a few are usually carried out.

If viewed as a PDF the headers of the different boxes are links that will take you to the relevant guide, if viewed in print you will have to navigate to the page number next to the title in each box.

Further information

Within Workbench, it is always possible to press F1 to get help about the active window. This will open the relevant help page from our online wiki page.

For advanced features, data examples, etc. go to our main wiki page:

http://www.ags-cloud.dk/Wiki/Workbench

This main page can also be accessed from within Workbench by pressing File –> help.

See the last pages of this manual for a list of keyboard shortcuts in the workbench.

AGS Aarhus GeoSoftware

Creating a workspace

The first thing to do in any workbench project is to create the workspace. The workspace is the main project wrapper; a single workspace is often enough to handle an entire project. The workspace can contain geophysical data from different instruments, borehole information, maps, inversions, visualizations and basic geological interpretations.

A new workspace is created by opening Aarhus Workbench and pressing “New” as seen in Figure 1.

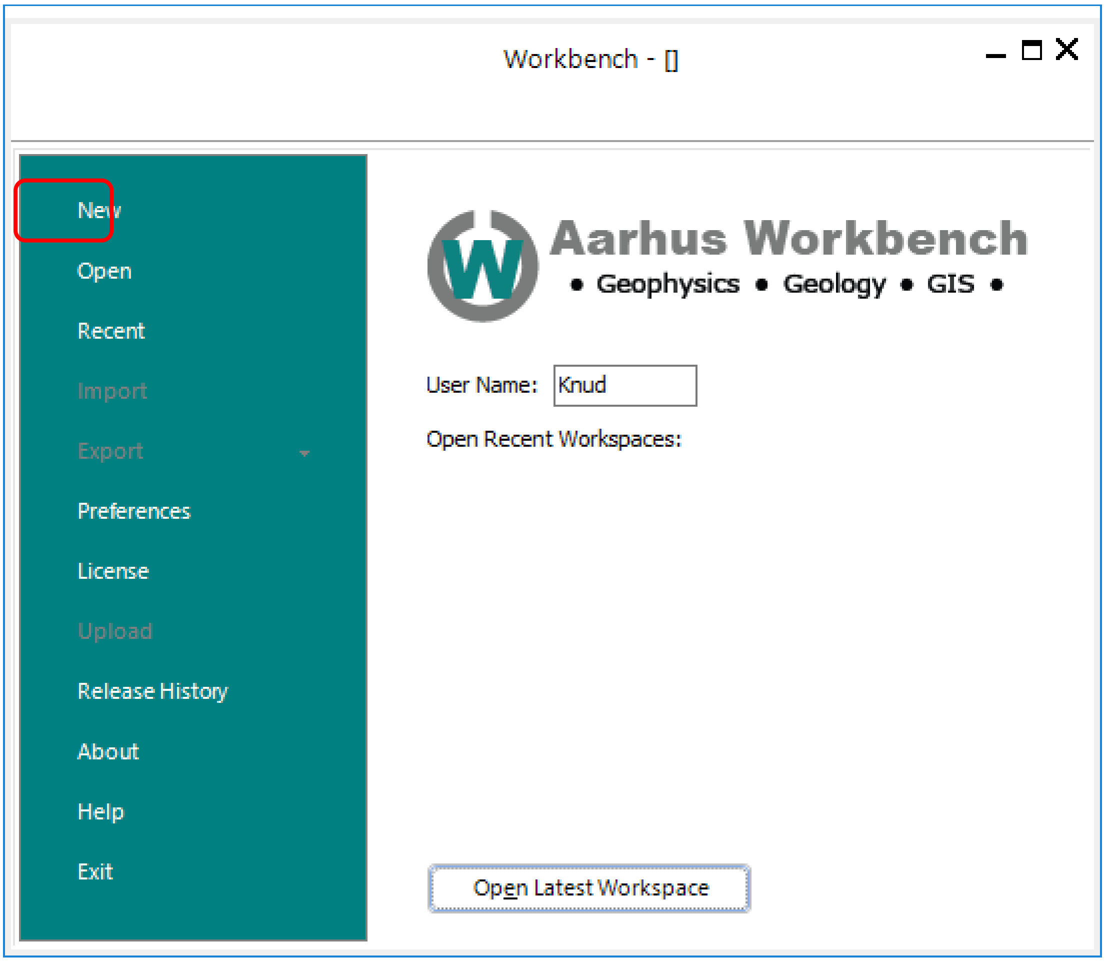


Figure 1 - Creating a new workspace

The wizard seen in Figure 2 will appear. In this window the name and location on the PC of the workspace is defined, the name and coordinate system of the map node in the workspace must also be defined. It is important to select a suitable coordinate system for the workspace e.g. to select the right UTM zone if UTM coordinates are used. If some of the data sets are in another coordinate system than the map node, the coordinates can be transformed on import.

It is also possible to select whether to go directly to the data import wizard right after creating the workspace, if you do not wish to do so just remove the checkmark in the “Open data import wizard” checkbox.

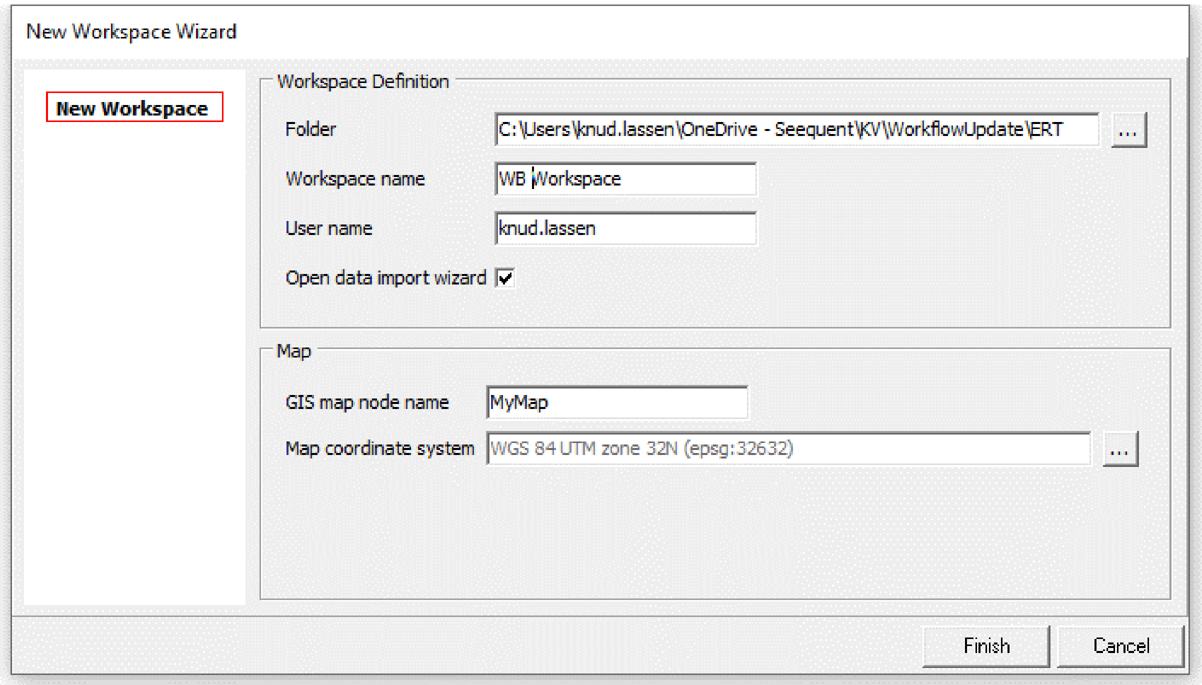


Figure 2 - Setting up the workspace

When pressing finish the workspace is created and opened, and the screen seen in figureFigure 3 is displayed, you are now ready to add a background map or start importing data into your workspace.

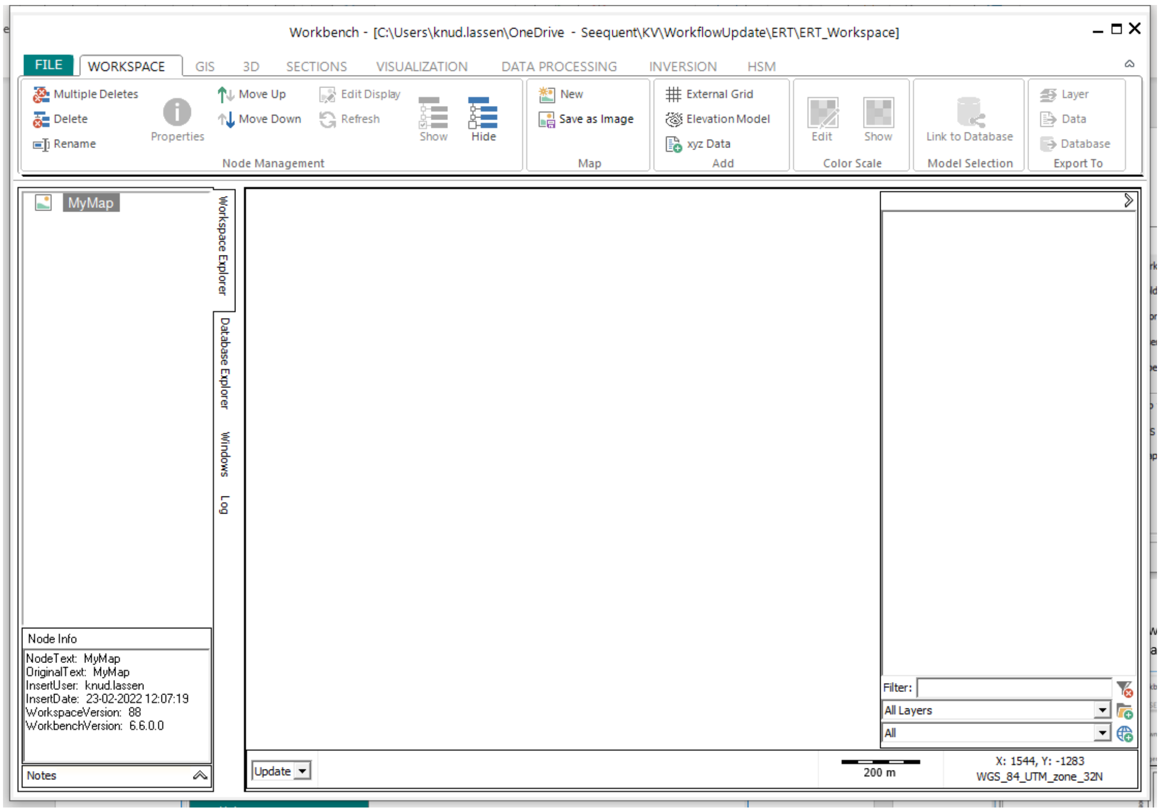
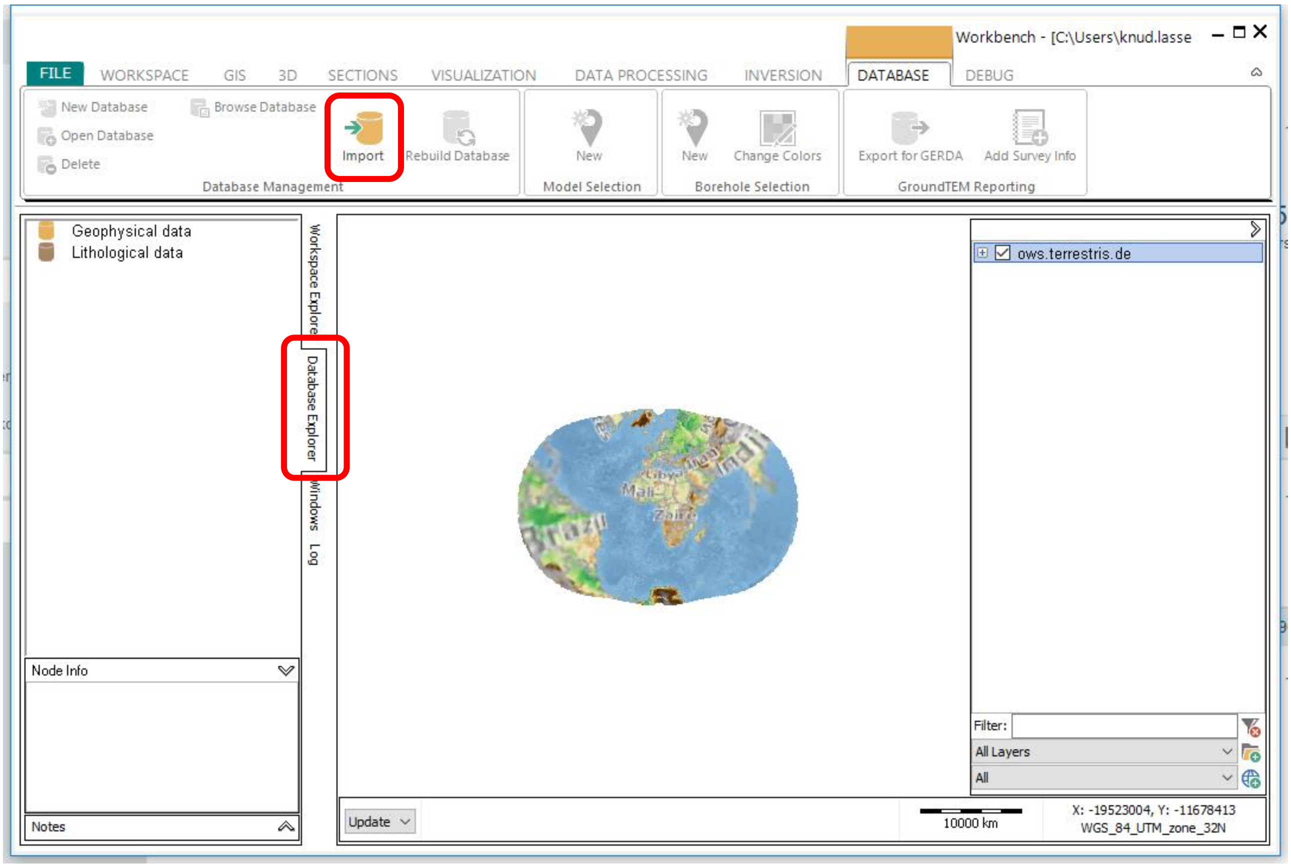


Figure 3 - Initial view of Aarhus Workbench after creating a workspace

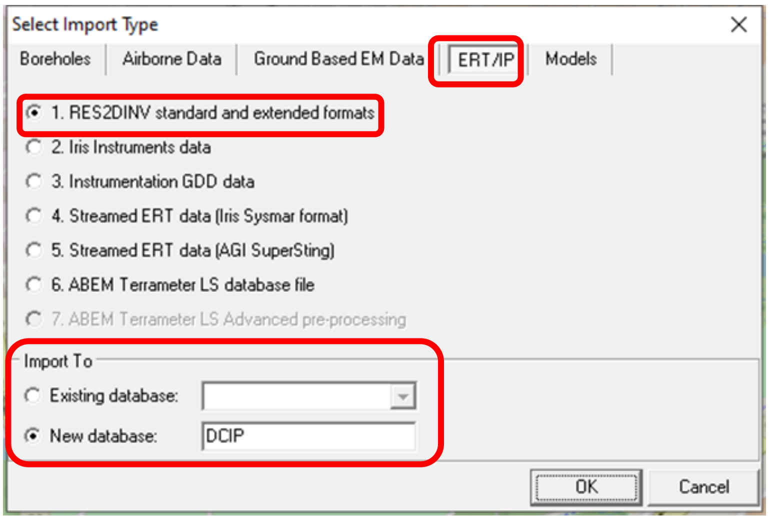
Data import

Data import is initialized by selecting the "database explorer" ribbon and pressing "import":

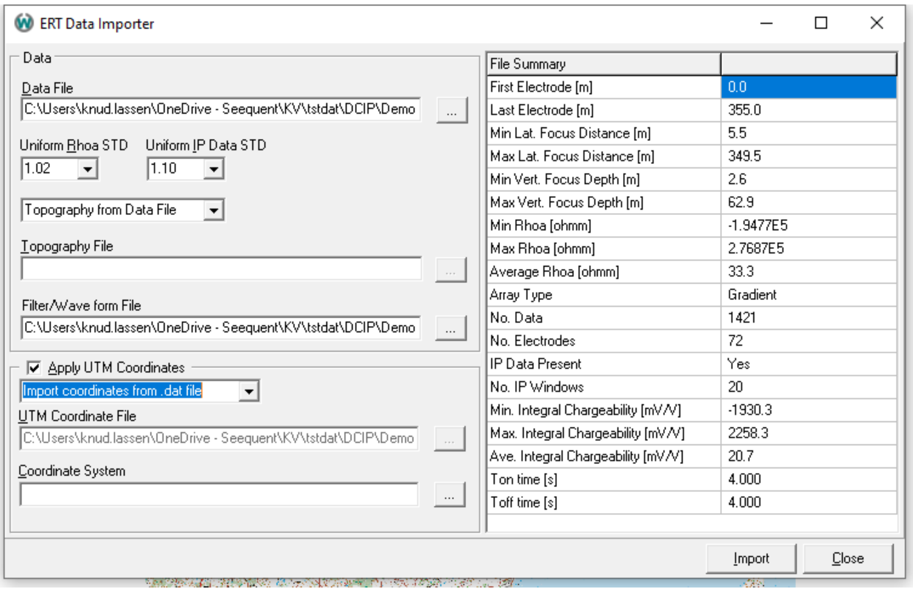


This opens the import window in which "ERT/IP" data is selected followed by "RES2DINV standard and extended formats". If importing in another format that the extended .dat format please see http://www.ags-cloud.dk/Wiki/W_GuidesERTIP for import guides.

If this is the first dataset in the workspace "New database" is selected and a name for the database is entered, if this is not the first import in the workspace an existing database can be selected.



When pressing OK the “ERT data importer” window is shown:



In this window the different files for the import can be selected. For DC-only measurements the .dat file is the only mandatory file, for DCIP measurements the filter/waveform (.ini) file is also mandatory. The .dat file is the main data file containing the measured DC and IP data, and the the .ini file contains information about the filters in the instrument and the transmitted waveform/current-pulse, the format of both the .dat and the filter/waveform file can be seen below:

Format of the .dat file (without coordinates and topography)

The .dat file is the data file, following the RES2DINV format but extended to accommodate the full waveform IP decay. For e.g. the ABEM Terrameter LS and IRIS Syscal, this .dat file can be exported directly from the instrument. An example of a .dat file is listed below including comments for each line. The format exists in two versions identified by the number of values in line 13. For further specification of array types etc. please refer to the RES2DINV manual (https://www.aarhusgeosoftware.dk/res2dinv-res3dinv).

Line no.	**Line text**	**Comments**
1	box1_1_1	Header (anything can be put here)
2	5	Electrode spacing in m
3	11	Array type
4	15	Sub-array type
5	Type of measurement (0=app.resistivity,1=resistance)	Explanation for line 6
6	1	Resistance (line 5)
7	763	No. of data points
8	2	Type of x-location. 2 for surface distance. 1 for true horizontal distance.
9	11	Flag for IP data
10	Chargeability	Explanation for line 11
11	mV/V	IP unit
12	12 0.02 0.02 0.04 0.06 0.08 0.1 0.14 0.18 0.26 0.4 0.6 0.88 1.2 4 4	Number of windows, delay time, width of each window, current ontime, current offtime.
13 and forward normal	4 190 0 280 0 230 0 240 0 0.066730629 17.03322055 14.92100834 12.81968784 11.15426881 9.887454957 8.780022266 7.745503221 6.904440409 5.477856057 4.262388796 3.325589266 2.770956927	Number of electrodes used, x and z location of C1, C2, P1, P2, apparent resistivity or resistivity value, IP value for each window.

13 and forward Extended	4 190 0 280 0 230 0 240 0 0.066730629 17.03322055 14.92100834 12.81968784 11.15426881 9.887454957 8.780022266 7.745503221 6.904440409 5.477856057 4.262388796 3.325589266 2.770956927 0.02 0.1 0.1 0.1 0.1 0.1 0.1 0.1 0.1 0.1 0.1 0.1 0.1 1 0 0 0 1 1 1 1 1 1 1 1 1	Number of electrodes used, x and z location of C1, C2, P1, P2, apparent resistivity or resistivity value, IP value for each window, std of apparent resistivity or resistivity, std value for each IP window, flag for DC, flag for each IP window.
Last line	0	Indicates end of file

Format of the .ini filter waveform file

The .ini file contains information about the filters in the instrument and the transmitted waveform/current-pulse and must be created manually. The .ini file is only required for IP data. Below is an example of a .ini file with settings from the ABEM Terrameter LS. Most of these settings are the same for each measurement, only line 4 – 8 needs to be changed if settings on the instrument are changed.

Line no.	**Line text**	**Comments**
1	[Waveforms]	Waveform section
2	NWaveForms=1	No. of waveforms. For now, always 1.
3	WaveTypes=1	Wavetype. For 50% duty cycle the WaveTypes=1, for 100% duty cycle it has to be WaveTypes=4.
4	NPulses=4	No. of pulses. Equal to no. of stacks *2. The number of stacks is set on the instrument.
5	StartDCInt=3.5	Acquisition delay. This is set on the instrument.
6	EndDCInt=4.00	Acquisition time + StartDCInt (line 5). Acquisition time is set on the instrument.
7	Ton=4, 4, 4, 4	Acquisition delay + acquisition time. One digit for each pulse (line 4).
8	Toff=4, 4, 4, 4	IP off time. This is set on the instrument. One digit for each pulse (line 4). For 100% duty cycle it has to be set to Toff= 0, 0, 0, 0
9	Amp=1, -1, 1, -1	Amplifier for each pulse. Sign reversal (positive, negative pulse)
10	[Filters]	Filter section. With these low pass and high pass filters you can model filters you know are present in the instrument and effect the data, it is best to leave this alone unless you are quite sure about what you are doing.
11	NFilters=0	
12	LPCutOffs=0	
13	LPOrders=0	
14	LPFreqs=0	
15	HPCutOffs=0	
16	HPOrders=0	
17	HPFreqs=0	

Including coordinates and topography in the data import

To take full advantage of all the GIS and visualization features of Aarhus Workbench it is necessary to include the location and topography of the dataset in the import. This can be done in a number of ways:

- The coordinates and/or topography of each profile can be included directly in the .dat file
- The coordinates and topography of each profile can be provided in a separate file (the .ewp file)
- The topography can be computed from an elevation model covering the area (in the .grd grid format), this requires that the coordinates are either included in the .dat file or in a .ewp file
- The topography can be provided in a separate file (the .eZ file)

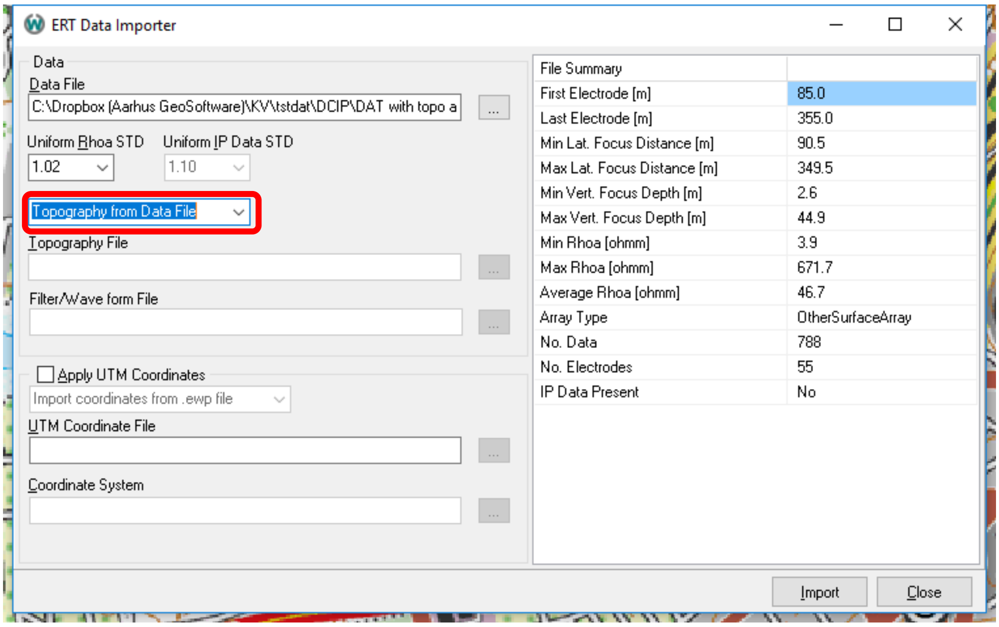
These ways are all equally good and can be combined in different ways depending on what suits the workflow of the user best. The most common options are to include both coordinates and topography in the .dat file, or to include the coordinates in the .dat file and deduce the topography data from a .grd elevation model. The different options are described in detail below:

Adding topography information to the .dat file

The format for including topography information in the .dat files is shown below, the topography points are added after the data section of the file. When including topography it is recommended to provide at least one topography point at each end of the profile and to provide sufficient points to describe all significant elevation changes along the profile, it is common to include one topography point for each electrode. NOTE: It is also possible to add the topography as the z coordinate of each electrode in the data lines.

Line no.	**Line text**	**Comments**
Last data line	4 190 0 280 0 230 0 240 0 0.066730629 17.03322055 14.92100834 12.81968784 11.15426881 9.887454957 8.780022266 7.745503221 6.904440409 5.477856057 4.262388796 3.325589266 2.770956927	Number of electrodes used, x and z location of C1, C2, P1, P2, apparent resistivity or resistivity value, IP value for each window.
1st topography line	Topography in separate list	Header for topography in separate list
2nd topography line	2	Header indicating what distances are used for the topography points x-coordinates. 1=true horizontal distances 2=distances measured along ground surface
3rd topography line	25	Number of topography points
4th topography line and forward	85, 17	X location along profile, elevation (one line for each topography point)
Last topography line	1	Number of topography point coinciding with first electrode
Last line	0	Indicates end of file

To import the topography values from the .dat file the option "Topography from Data file" must be selected from the dropdown as seen below.

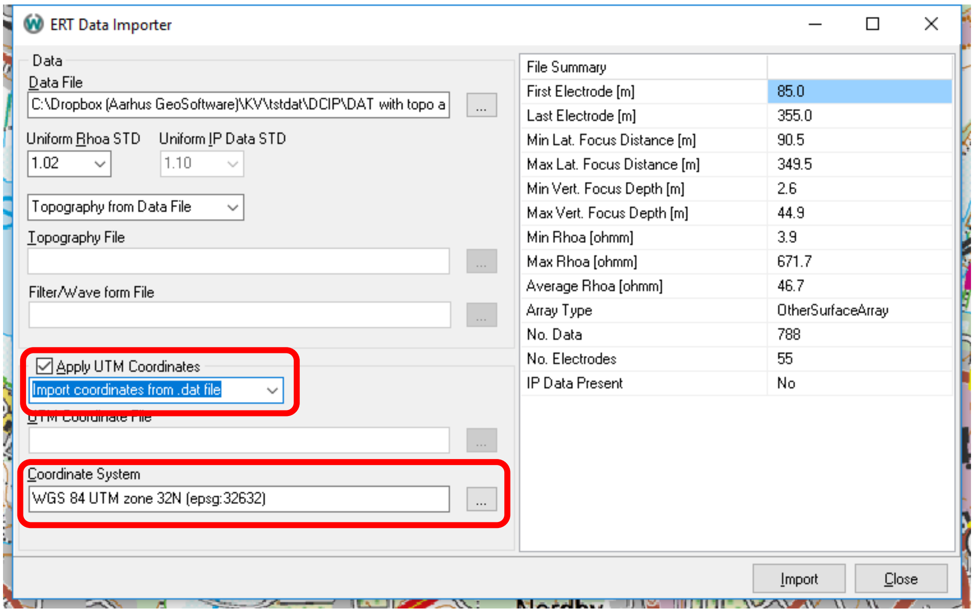


Adding global coordinates to the .dat file

The format for including global coordinates in the .dat files is shown below, the coordinates are added after the data section of the file. When including global coordinates at least two coordinate points must be provided, it is recommended to provide at least one coordinate point at each end of the profile and one point for each bend, if any, of the profile.

Line no.	**Line text**	**Comments**
Last data line	4 190 0 280 0 230 0 240 0 0.066730629 17.03322055 14.92100834 12.81968784 11.15426881 9.887454957 8.780022266 7.745503221 6.904440409 5.477856057 4.262388796 3.325589266 2.770956927	Number of electrodes used, x and z location of C1, C2, P1, P2, apparent resistivity or resistivity value, IP value for each window.
Last data line +1	0	Header indicating that topography is not included in the .dat file
1st coordinate line	Global Coordinates present	Header indicating that global coordinates are included in the .dat file
2nd coordinate line	Number of coordinate points	Header
3rd coordinate line	55	Number of coordinate points
4th coordinate line	Local Longitude Latitude	Header
5th coordinate line and forward	85.00 572088.12 6222426.57	X location along profile, longitude/UTMX, Latitude/UTMY (one line for each coordinate point)
Last line	0	Indicates end of file

To import the global coordinates from the .dat file check the checkbox “Apply UTM Coordinates”, select “import coordinates from .dat file” in the dropdown, and select the correct coordinate system in the “Coordinate System” box, as show below:



NOTE: The topography can also be added at the individual data lines of the .dat files, as a z-coordinate on each of the electrodes in each of the data lines (these values are set to zero in the above examples), this topography can also be imported by selecting “import coordinates from .dat file”. If topography is found both at the data lines and at the end of the .dat file the topography at the end of the file overrides the topography in the data lines.

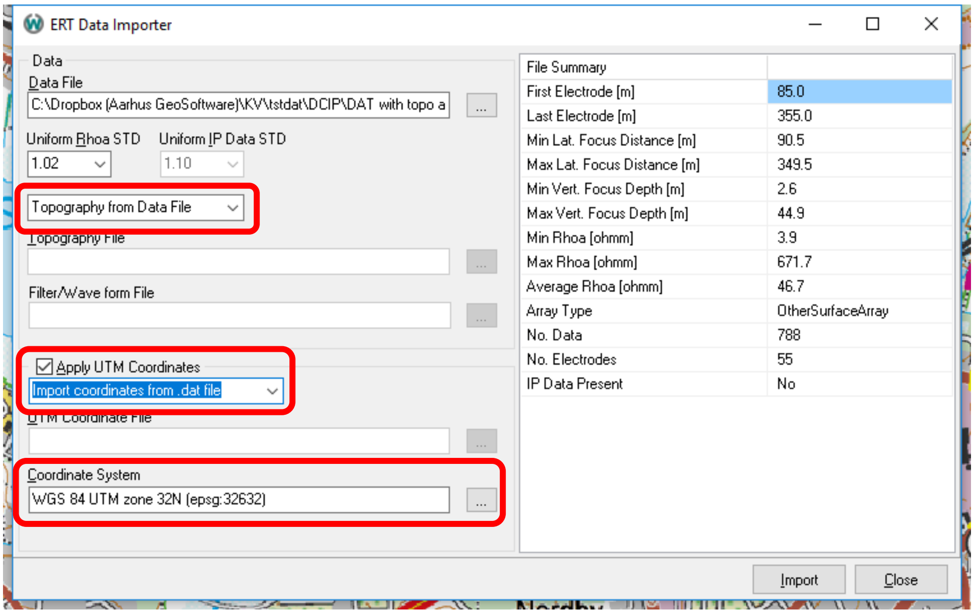
Adding both topography and global coordinates to the .dat file

The format for adding both topography and global coordinates is shown below, the topography section must always come before the coordinate section:

Line no.	**Line text**	**Comments**
Last data line	4 190 0 280 0 230 0 240 0 0.066730629 17.03322055 14.92100834 12.81968784 11.15426881 9.887454957 8.780022266 7.745503221 6.904440409 5.477856057 4.262388796 3.325589266 2.770956927	Number of electrodes used, x and z location of C1, C2, P1, P2, apparent resistivity or resistivity value, IP value for each window.
1st topography line	Topography in separate list	Header for topography in separate list
2nd topography line	2	Header indicating what distances are used for the topography points x-coordinates. 1=true horizontal distances 2=distances measured along ground surface
3rd topography line	25	Number of topography points
4th topography line and forward	85, 17	X location along profile, elevation (one line for each topography point)
Last topography line	1	Number of topography point coinciding with first electrode
1st coordinate line	Global Coordinates present	Header indicating that global coordinates are included in the .dat file
2nd coordinate line	Number of coordinate points	Header
3rd coordinate line	55	Number of coordinate points
4th coordinate line	Local Longitude Latitude	Header
5th coordinate line and forward	85.00 572088.12 6222426.57	X location along profile, longitude/UTMX, Latitude/UTMY (one line for each coordinate point)
Last line	0	Indicates end of file

To import both topography and global coordinates from the .dat select the “Topography from Data file” option from the first dropdown, check the checkbox “Apply UTM Coordinates”, select “import coordinates from .dat file” in the second dropdown, and select the correct coordinate system in the “Coordinate System” box, as show below:

AGS Aarhus GeoSoftware

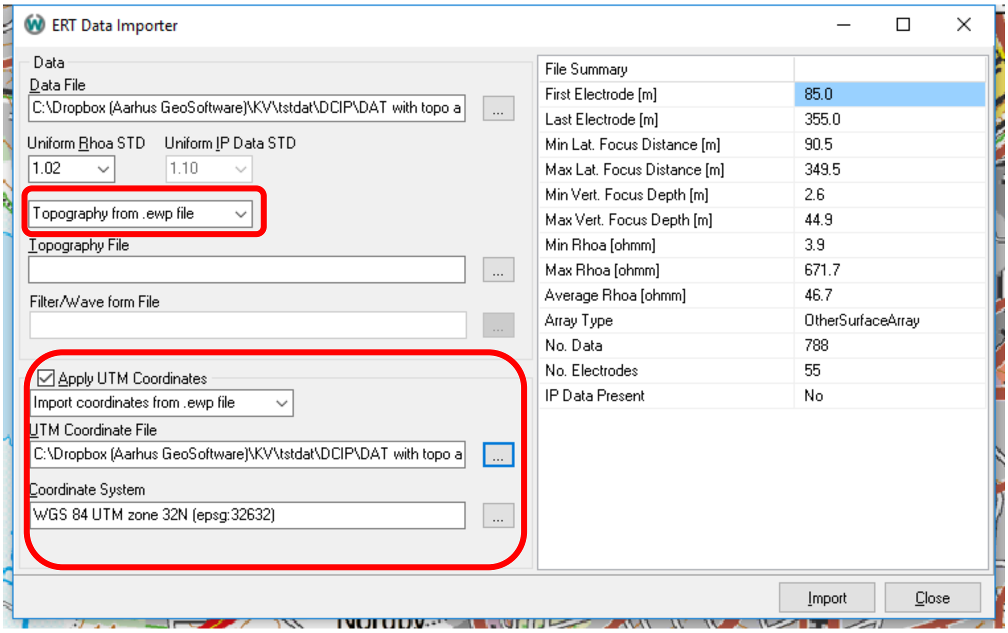


Importing topography and global coordinates from the .ewp file

The format of the .ewp files is seen below. Note that the first line is a header and that all other lines follow the same format:

Line no.	Electrode no.	utm x coordinate	utm y coordinate	Standard deviation (m) for x coordinate (GPS precision)	Standard deviation (m) for y coordinate (GPS precision)	Electrode distance (m)	Standard deviation (m) for electrode distance	Flag if profile makes turns	Elevation.
1	No	Utmx	utmy	stdx	stdy	edist	Stde	Angcon	utmz
2	1	494940.17	6177505.91	3	3	5	0.1	0	3
3	83	494775.71	6177129.36	3	3	5	0.1	0	30

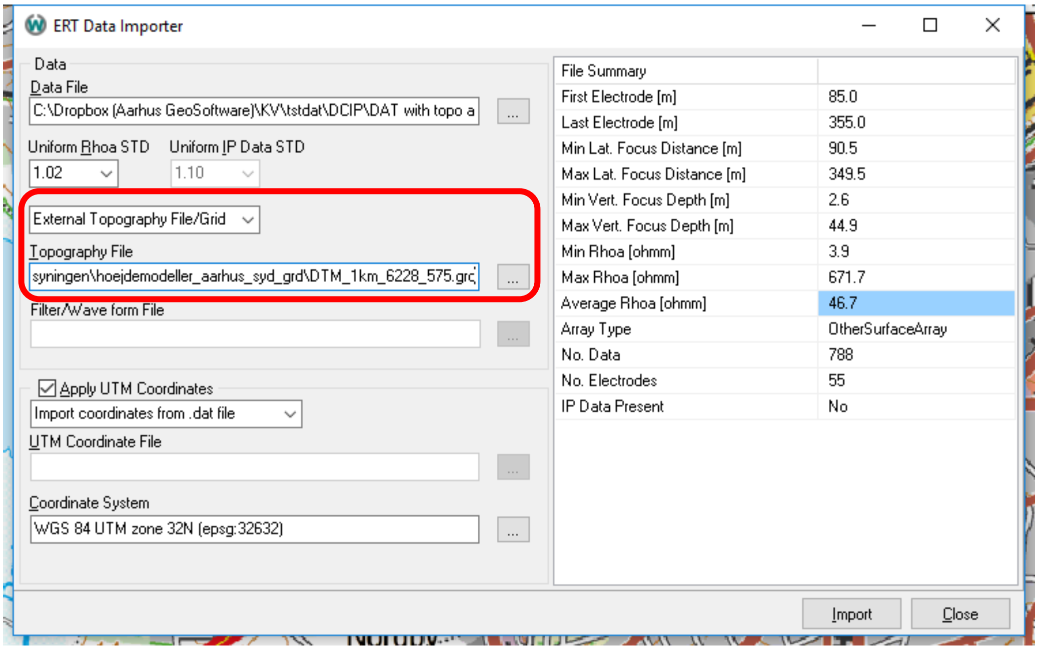
To get both coordinates and topography from the .ewp file select the "Topography from .ewp file" option from the first dropdown, check the checkbox "Apply UTM Coordinates", select "import coordinates from .ewp file" in the second dropdown, select the .ewp in the "UTM Coordinate File" field, and select the correct coordinate system in the "Coordinate System" box, as show below:



Note that it is also possible to only import the coordinates from the .ewp file and to get the topography from another source by simply selecting another option in the first dropdown menu.

Extracting the coordinates from an elevation model

If a sufficiently detailed digital elevation mode (DEM) covering the investigation are exists, this can be the easiest option for applying topography to the dataset, as the topography doesn't need to be added to a separate file for each data profile. The DEM must be in the .grd format (Golden Software binary grid format, it is easy to convert most elevation models to this format using a GIS software package e.g. QGIS). To use this option it is necessary to import the coordinates from either the .dat file or an .ewp file, to select "External Topography File/Grid" from the first dropdown, and to specify the .grd file in the "Topography File" field:



Importing topography from the .Ez file

Finally the topography can be imported from a .eZ file, the format of the .eZ file is shown below:

Line no.	Line text	Comments
1	Topographical data	Header
2	2	Type of x-location. 1 = true horizontal distance, 2 = distance along the ground surface.
3	156	Number of coordinates
4	100 -1.65	First X and Z coordinates of topography point along the profile
5	200 -0.49	Second X and Z coordinates of topography point along the profile
6	…	Remaining X and Z coordinates of topography point along the profile
7	10000 4.94	Last X and Z coordinates of topography point along the profile
8	1	The topography data point where the first electrode is written. Here the first electrode is positioned in horizontal coordinate of 100.

The use this import select "External Topography File/Grid" from the first dropdown and specify the .eZ file in the "Topography File" field. The .eZ file can be used without specifying any coordinates.

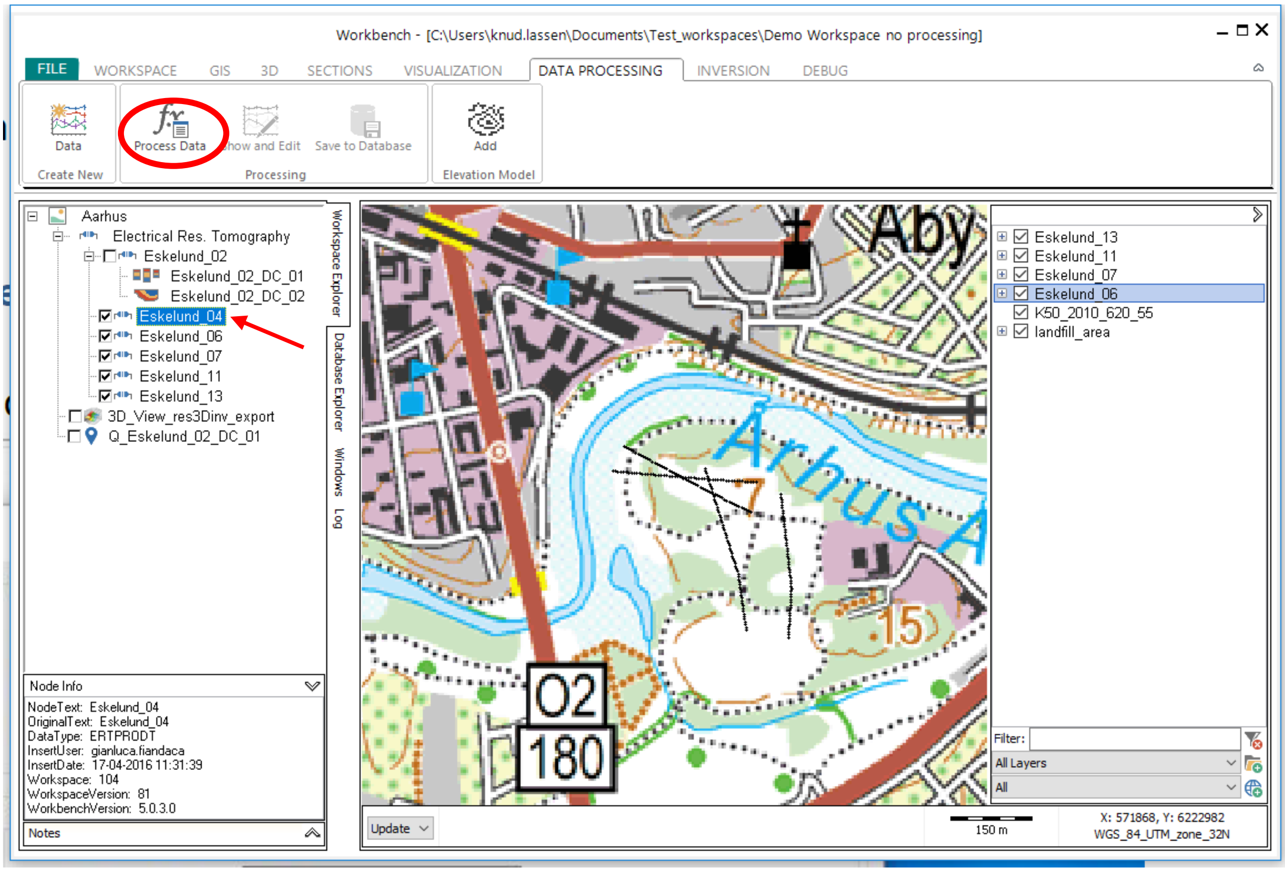
Guide for basic processing of ERT data with full waveform IP data

The workflow presented in the following will give good results for most full waveform DC/IP datasets. For some data sets, with low noise levels and no couplings, good results can be obtained with far less or even no processing, contrary. If the data are of too bad quality no amount of processing can give perfect results.

As a supplement to this guide a detailed description with data examples of the manual processing of DC and IP data can be found in the video tutorials here: http://www.ags-cloud.dk/Wiki/W_ERTVideoTurtorials

Introduction

After completing the data import the processing is initiated by selecting the processing node in the workspace explorer and pressing the “Process Data” option in the “DATA PROCESSING” ribbon.



Initial DC processing

The first step in the processing is to ensure a well fitted DC mode. The main tools in the DC data processing are the pseudo section and the data profiles.

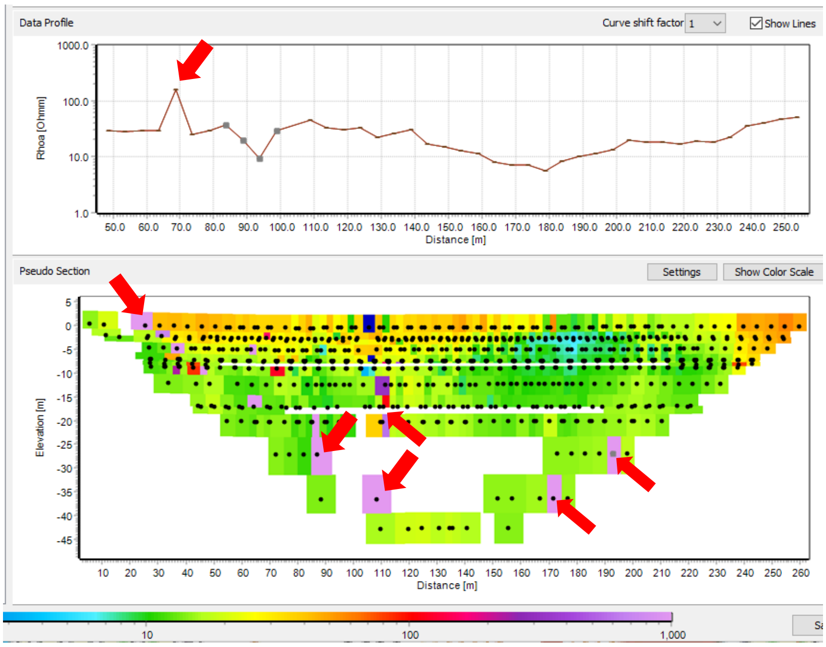


Figure 4

In the pseudo section the apparent resistivity is plotted on a color scale for all electrode configurations, the data profile shows the same data but plotted as a graph for each focus depth, the different focus depths can be selected in the window to the left of the profile and pseudo section.

The task is to remove obvious outliers in the data as exemplified by the red arrows in the above figure. These can be due to bad electrode contact, instrument malfunction etc.

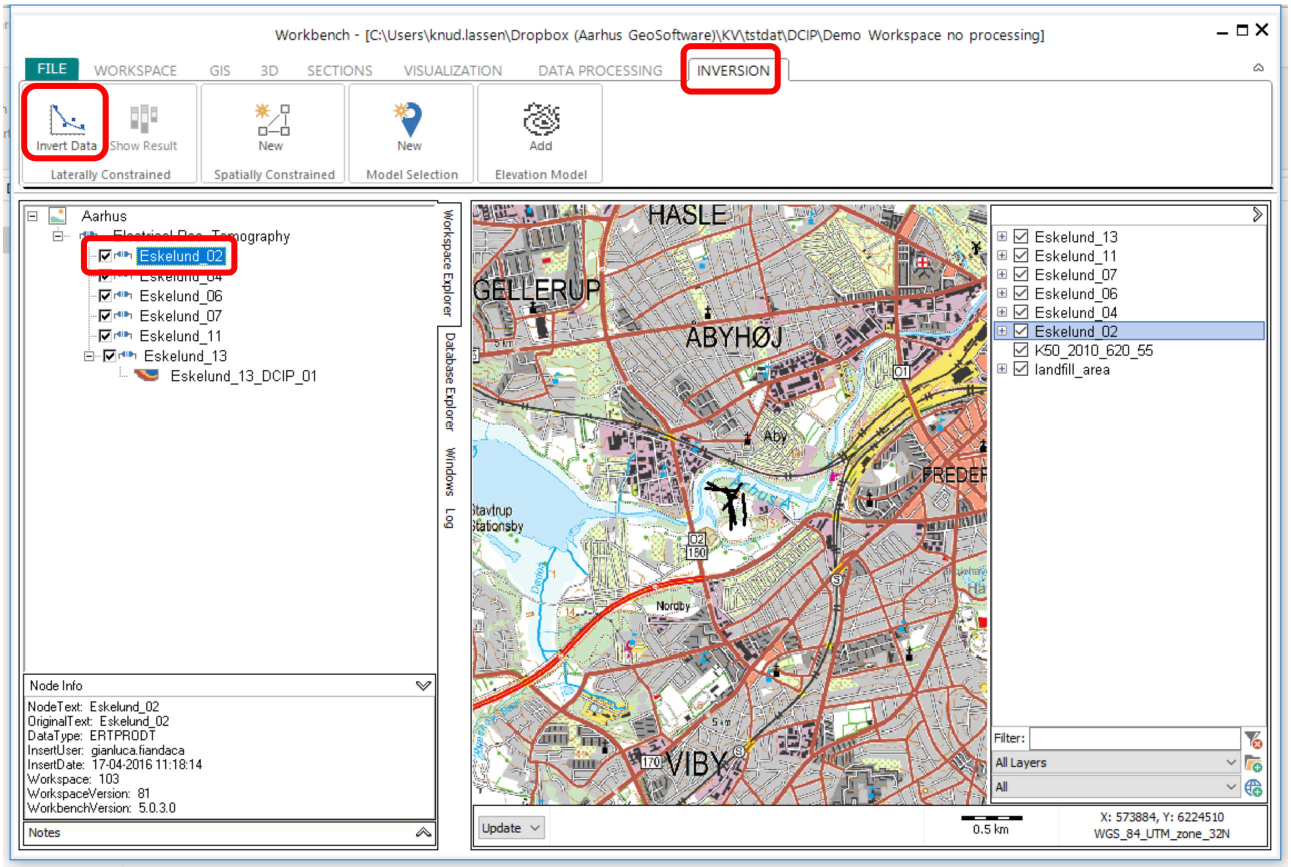
Data points are toggled on and off by pressing **Alt+A** and **Alt+Q** while the points are selected, it is also possible to remove all datapoints in which a bad electrode is involved (e.g. due to bad contact or faulty cables) by selecting the electrode in the "Electrode Positions" window.

DC inversion and reprocessing

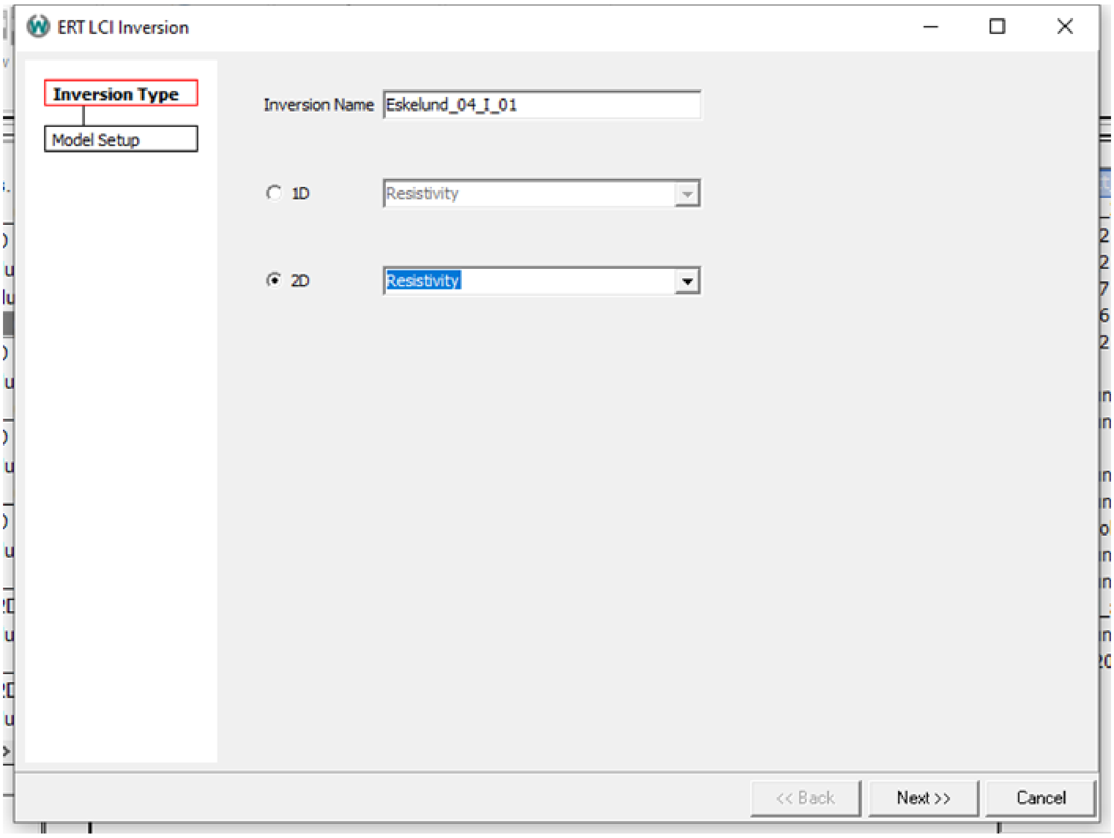
The next step is to run a DC inversion and evaluate the data fit using the DC residual pseudo section. This is primarily a task of either removing or assigning a higher standard deviation to data points with a high misfit, and to evaluate whether the chosen constraints and model discretization is suitable for the geological setting.

Running a DC inversion

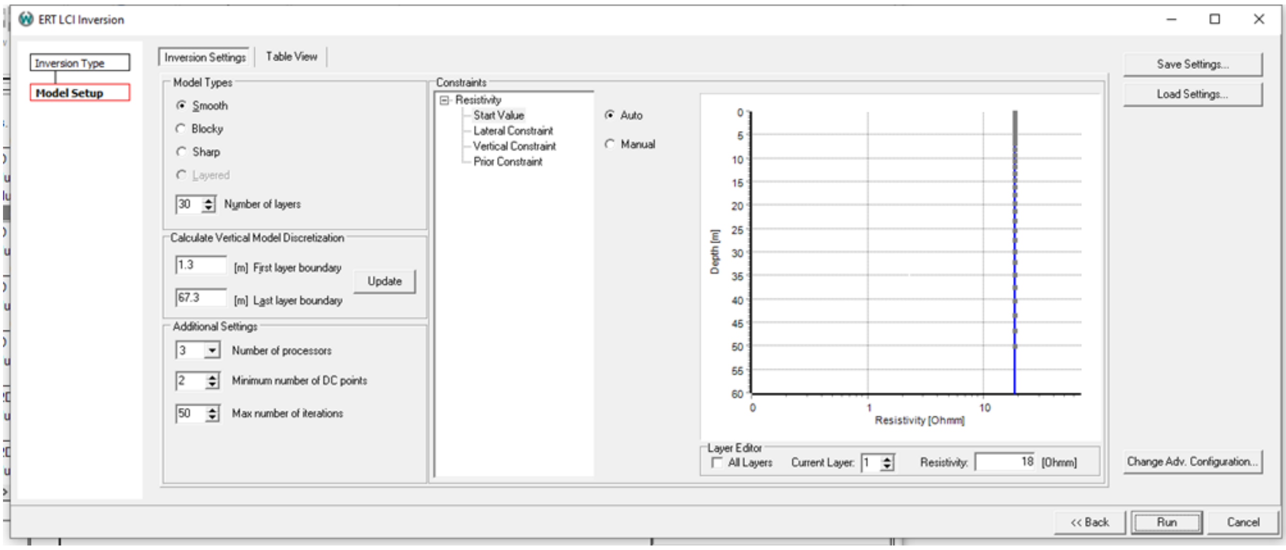
A DC inversion in run by highlighting the inversion node and pressing "Invert Data" in the "INVERSION" ribbon:



This opens the following dialog box in which the type of inversion is selected:

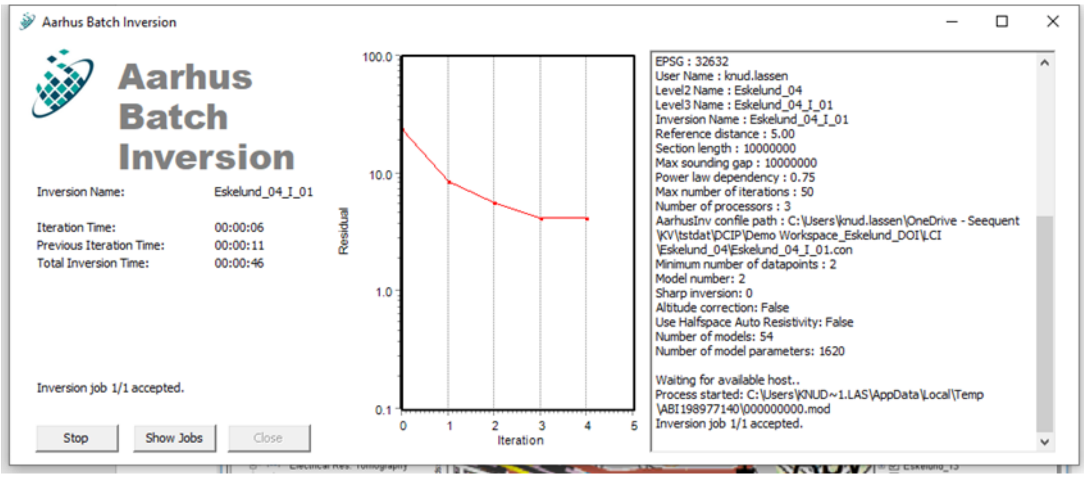


In this case a 2D resistivity inversion is used as the basis for further processing. Selecting “Run” will open a new window asking for the name of the inversion after which a second window will open in which a number of inversion settings can be specified:



For the initial inversions a smooth inversion with default settings will be a good starting point in most situations. Remember to click on “Update” to use the suggested vertical model discretization. The “Update” button is grey if the displayed values are used. Otherwise the values from the last inversion is used.

When selecting “Run” the inversion will start, the progress can be tracked in a separate window:



It is possible to continue to work in the workbench while the inversion runs, the inversion result is automatically imported into the workspace once the inversion is done and the inversion window can then be closed.

Reprocessing based on inversion results

Once an inversion has been carried out the result can be seen by highlighting the inversion in the “Workspace Explorer” and selecting “Show Result” in the “INVERSION” ribbon. The default way to display results is as a resistivity model and corresponding residual pseudo section.

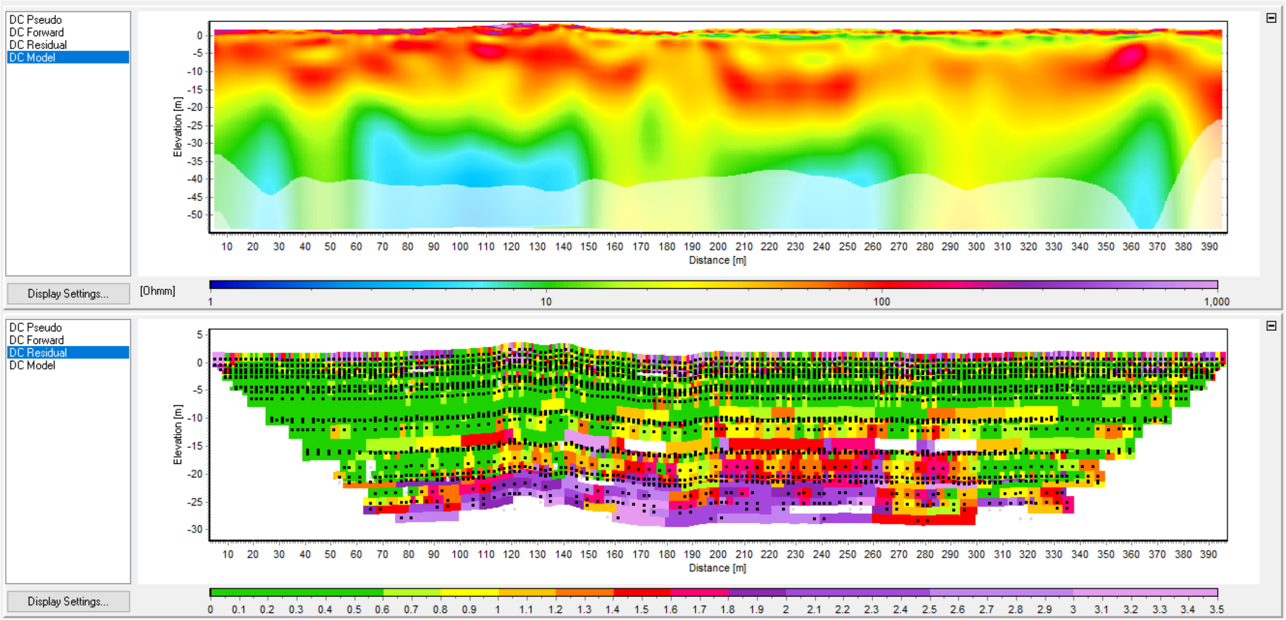


Figure 5 - total residual = 1.11, data example kindly provided by VIA University College, Horsens, Denmark.

Seen above is an inverted resistivity model with corresponding DC residual pseudo section with a very high misfit for the deepest configurations, in this case the problem was due to a too shallow discretization of the model, by increasing the depth to the deepest layer from 45 to 60 meters (for a 20-layer smooth model) the misfit was drastically reduced as seen in the pseudo section below.

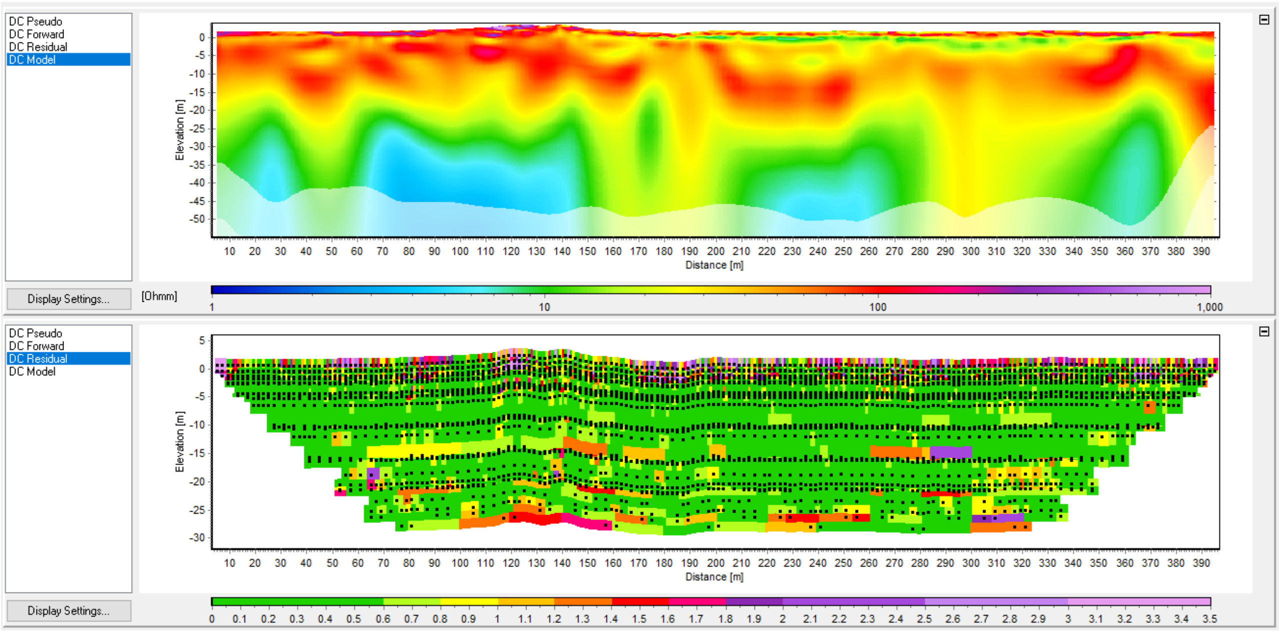


Figure 6 - total residual = 0.86, data example kindly provided by VIA University College, Horsens, Denmark.

Finally increasing the standard deviation of the very surface near, noisy data points to 10% (by selecting the points and pressing **Alt+2**) and disabling a few of the worse fitting points the misfit was further reduced as seen below in the final inversion result.

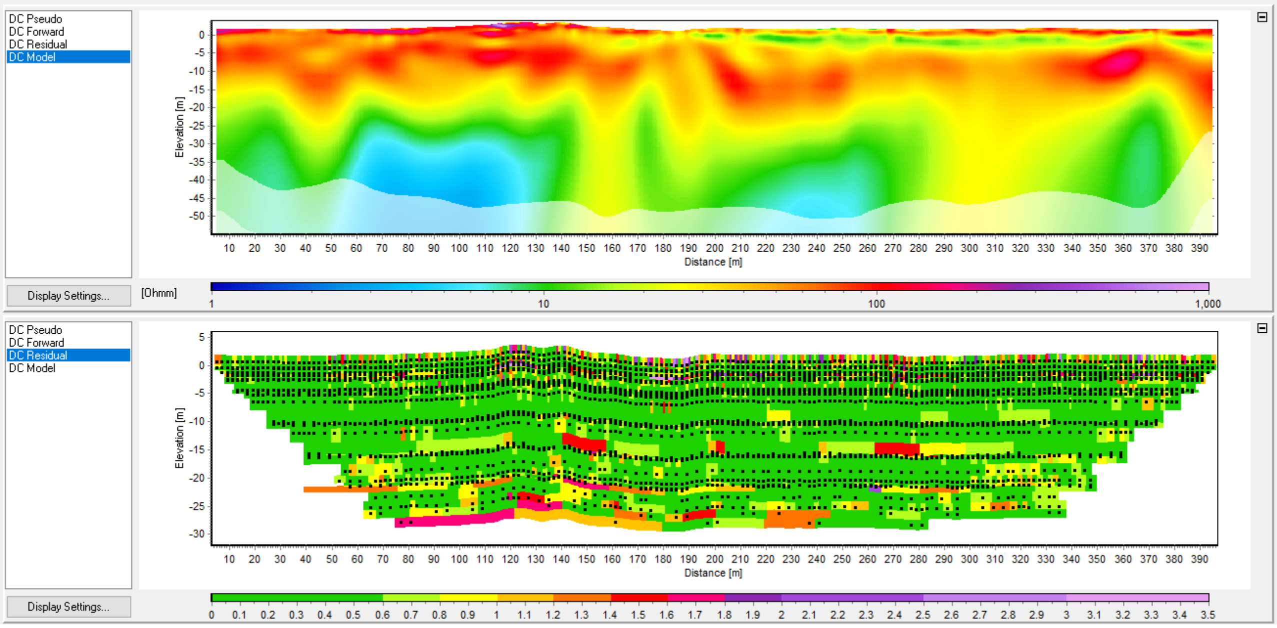


Figure 7 - total residual = 0.59, data example kindly provided by VIA University College, Horsens, Denmark.

Note that data points can be disabled directly from the DC residual pseudo section in the inversion result window, by selecting the data point and pressing **Alt+Q,** and that if both an inversion result window and a data processing window is open at the same time selections are synchronized between the two, so that it e.g. is possible to see which electrodes are involved in the measurement of a poorly fitting data point**.** After obtaining a satisfactory DC processing and inverted model the next step is processing of the IP data.

IP processing and Inversion

After processing the DC data and making sure a good data fit is obtained it is time to move on with the IP processing. The IP inversion depends on the DC data, so there's no reason to try to process the IP data before the DC processing is done, as all IP data points are automatically disabled for a configuration when the DC data point is disabled.

Setting up noise model

First step is modifying the noise model for the IP data, due to the, often low, signal to noise ratio of the IP decay data, the uniform 5% standard deviation often applied to data during import is not representative for the actual data uncertainty. The noise model can be modified in the processing menu as seen in Figure 8 below.

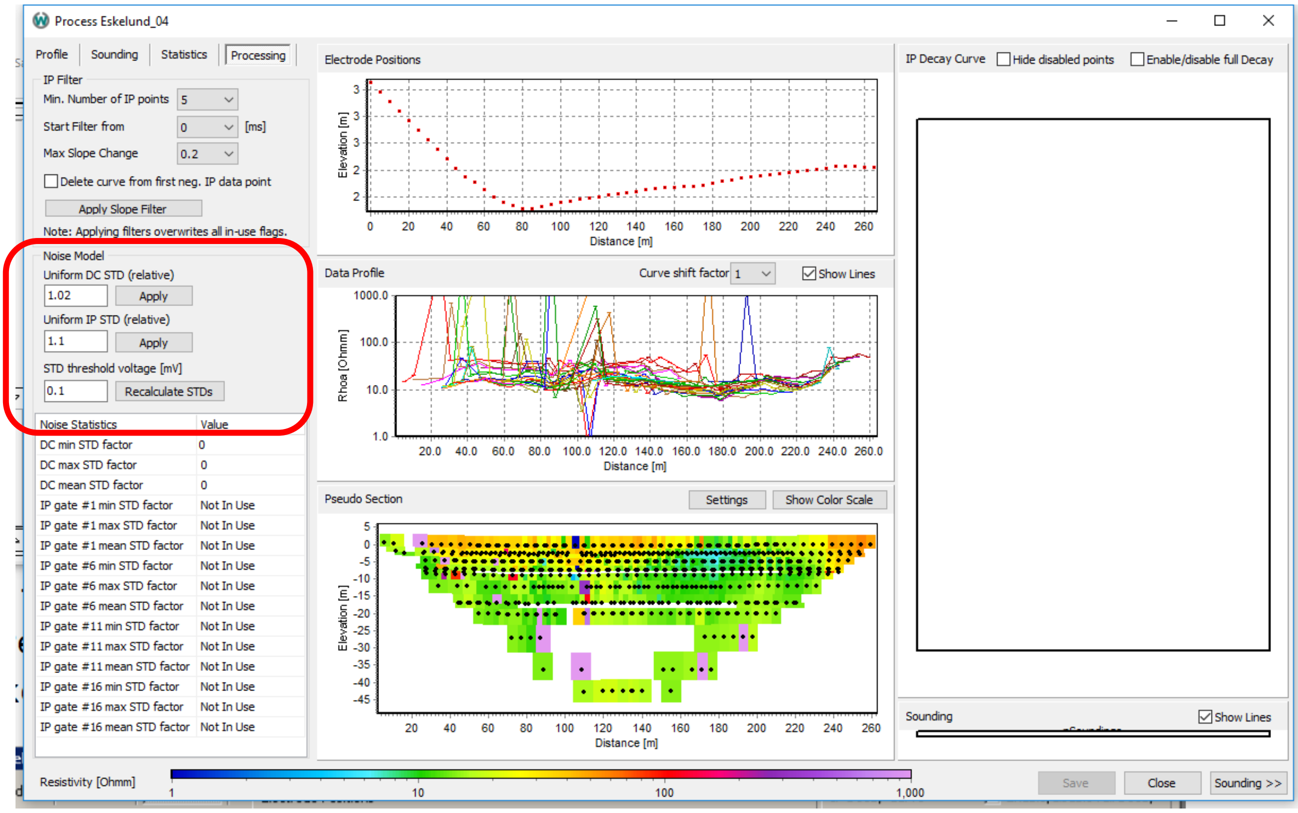


Figure 8

A uniform STD of 1.1 (10%) is often suitable, the STD threshold setting makes data points closer to this threshold have a larger STD, helping the inversion converge regardless of noisy decays. The new noise model is applied by clicking “recalculate STDs”. If very noisy IP decays are found during processing the standard deviation/error bars can be made bigger by selecting the points with the mouse and pressing **Alt+2** (for 10% standard deviation), **Alt+3** (for 15% standard deviation) or **Alt+4** (for 20% standard deviation) etc. For a full list of processing keyboard shortcuts see: http://www.ags-cloud.dk/Wiki/WH_KeyboardShortcuts. If a lot of decays needs additional uncertainty it can be a good idea to revisit the noise model and increase either the uniform IP STD or the STD threshold voltage.

Automatic processing/filtering

In the automatic IP processing it is possible to filter out noisy or faulty IP decays based on the slope change, this function can remove a lot of bad decays, but is rarely enough that no manual processing is needed. It is possible to only filter based on data from 100ms and onwards to avoid filtering on effects caused by lowpass filters in the instrument. Figure 9 - Noisy IP decays removed by the slope change filterFigure 9 shows an example of the type of decays that can be removed using the filter. The settings for the IP slope filter can be found above the settings for the noise models as seen in Figure 8. It is also possible to make the filter remove all negative decays by checking the “Delete curve from first neg. IP data point” box.

This filter should only be applied once in the beginning of the IP processing phase, as all previous IP processing for the line is overwritten in the process.

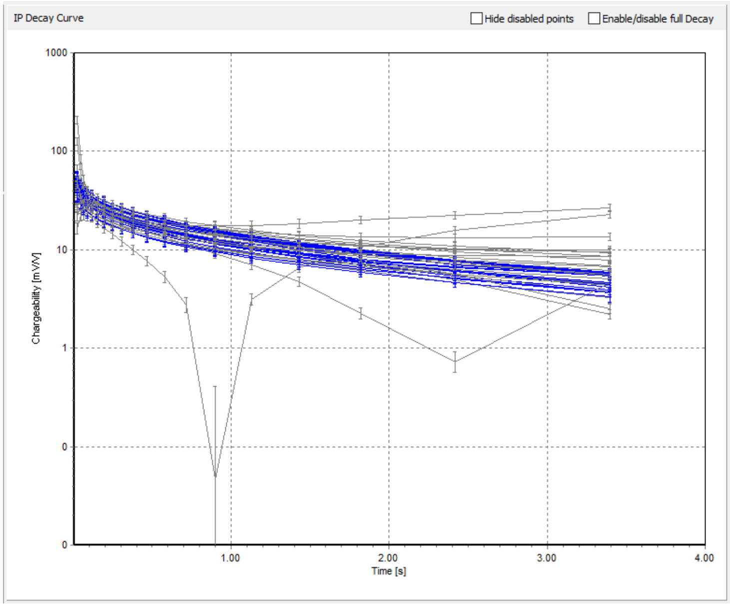


Figure 9 - Noisy IP decays removed by the slope change filter

Manual processing of full waveform IP data

The automatic IP processing should be followed up by a manual processing. Again, this is a question of removal of outliers. To make it easier to spot potential outliers it is possible to change the pseudo section and data profile to show one of the IP decay windows instead of the DC values, as seen in Figure 10, the change is made in the dropdown menu to the left in the data processing window. It can be seen that some decays have a gate 5 value that is clearly very different from the neighboring measurements, this is a good indication that something might be wrong with the entire decay. The main chore of the IP processing is to go through all the decays a few at a time and delete all unwanted decays. This is done by selecting a few (10-50) decays close to each other from the pseudo section or data profile. Due to the averaging nature of the DC/IP method these are not expected to look very different from each other, so outliers should be removed by selecting the decay and pressing **Alt+Q.** It is possible to either remove individual gates or entire decays by checking the checkbox at the top right of Figure 10, it is also possible to hide already disabled decays to get a uncluttered picture of the still in use data.

Deciding which decays to keep and which to discard is not easy, and requires patience, trial and error, and building experience. Figure 11 shows a few examples of IP decays removed during processing.

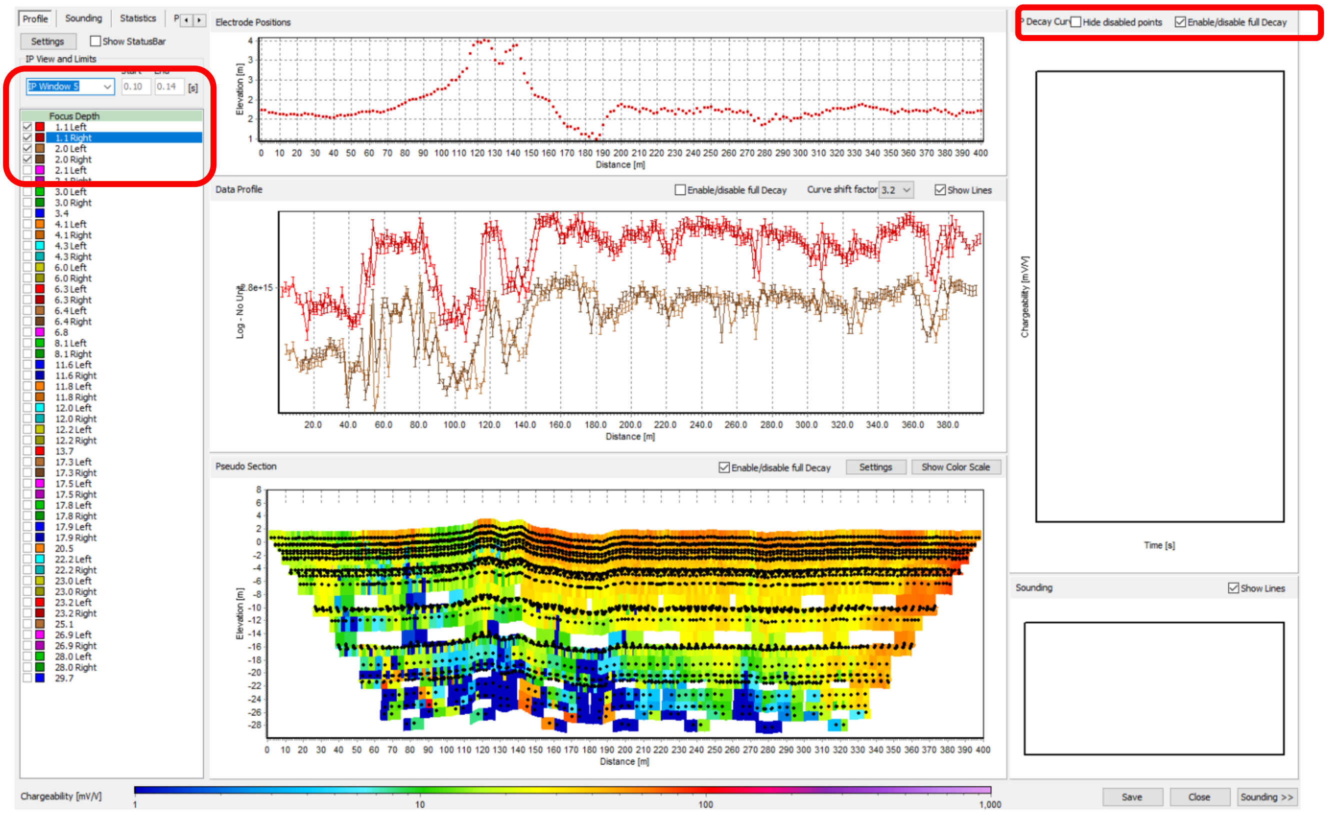


Figure 10 - IP Window 5 data profile and pseudo section, with only the most surface-near data points activated in the data profile, data example kindly provided by VIA University College, Horsens, Denmark.

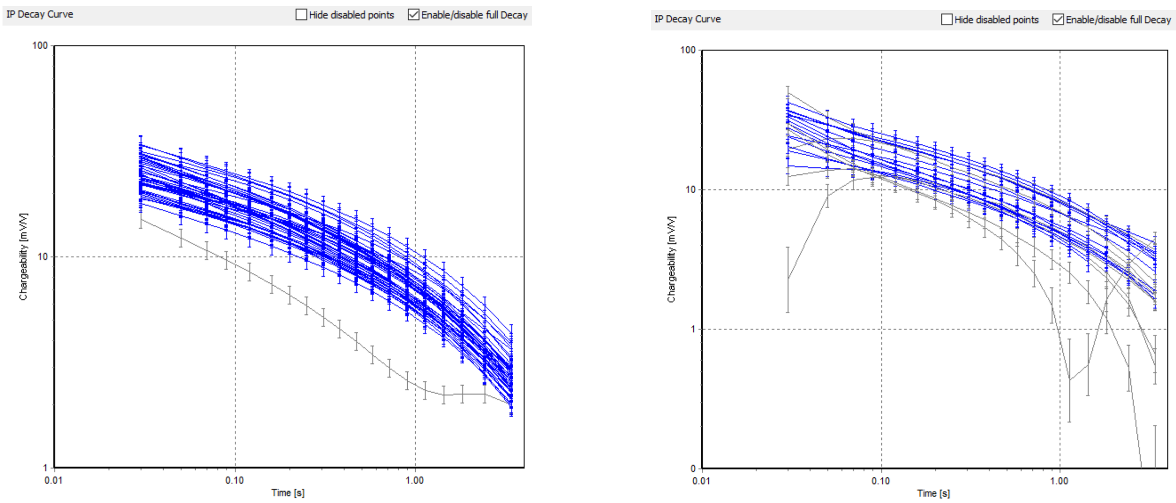


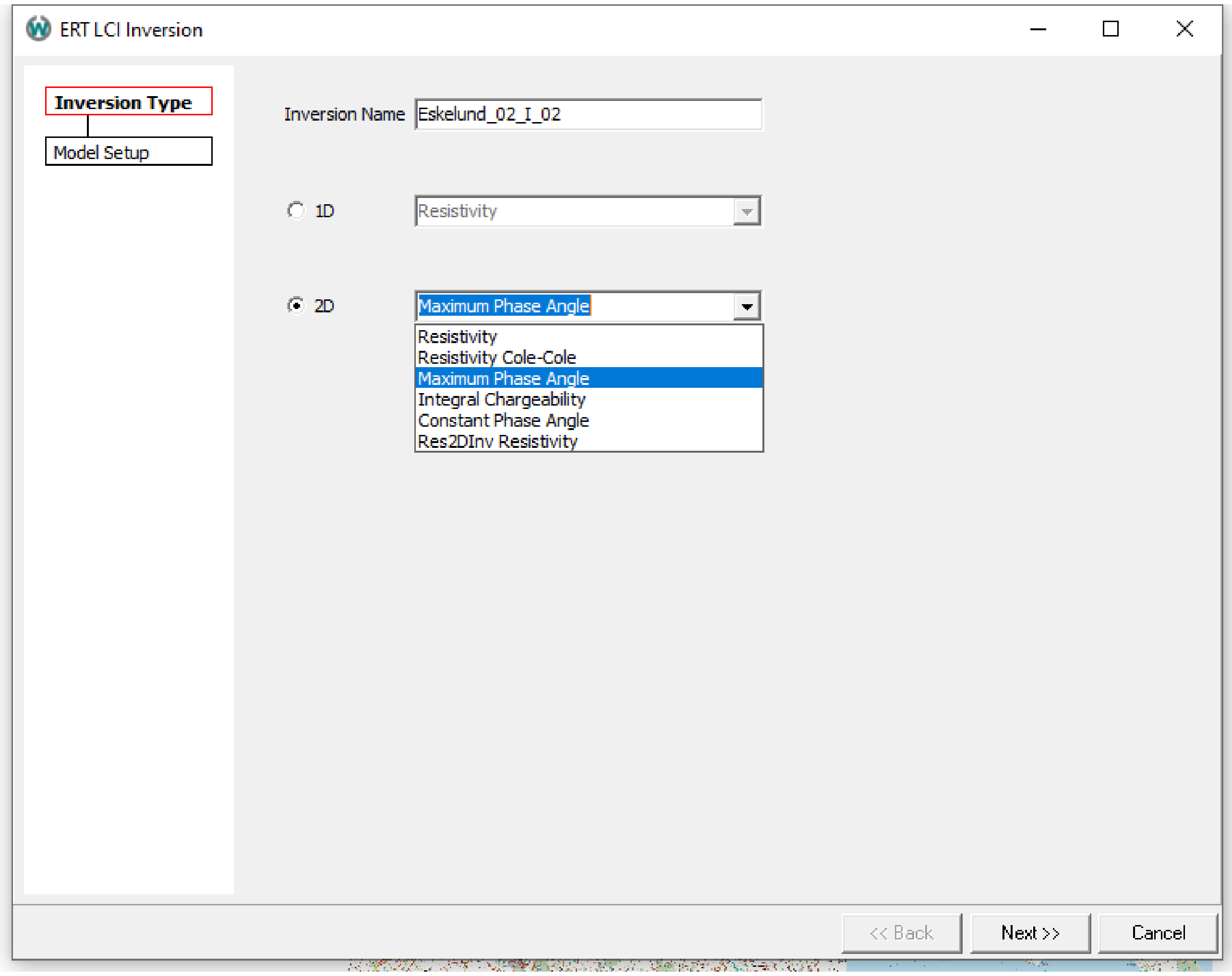
Figure 11 – (Left) Decay disabled due to different signal level compared to neighboring decays, and upward bending shape at late times, possibly due to drift removal. (Right) Decays disabled due to very different shape compared to neighboring decays.

Remember to save the processing before closing the processing window.

IP inversion and reprocessing

After the DC and initial IP processing is carried out it is time to run an inversion including the IP data.

This is done by selecting one of the IP parametrizations e.g. Maximum Phase Angle in the inversion setup:



The rest of the setup is very similar to a resistivity only inversion.

If the fit of the IP model is not satisfactory the procedure is the same as for the DC data, that is, evaluation of the ill-fitting data points, removal of outliers, evaluation of the model discretization, constraints and inversion type and finally, reinversion. Note that in the same way as with the DC data it is possible to reprocess the IP decays directly from the inversion result window, by selecting the point from the pseudo section and process the displayed decays.

When inverting with IP parameters, the residual for the DC data will often not be as good as when inverting for DC alone. This is because the inversion has more parameters and it will try to find the best compromise between the best fit between the DC and IP parameters.

It is also an option to further tweak the parameters for the IP noise model shown in figure 5 if the data fit is still not satisfactory, but note that this will overrule all previous IP processing.

After obtaining satisfactory inversion results for all datasets in a workspace it is time to move on with visualization, interpretation and presentation of results.

Adding a background map

Adding maps to your workspace is often a good tool. General maps or aerial photos are great for relating the location of data and models to the real world. And specialized maps, such as geological maps or maps showing powerlines and other infrastructure, can be valuable tools when processing data and interpreting inversion results. Both local and online maps can be added.

Local maps

Local maps are maps that are stored on your computer, many raster and vector formats are supported, such as: .tab, .mif, .shp, .tif, .jpg, .jp2, .ecw and .kml.

The layers are added by pressing the “open folder” button at the lower right corner (see Figure 12) and selecting the file in the window that opens. The different GIS/MAP layers can be toggled on and off in the GIS layer control to the right on the screen, it is also possible to edit the order of the layers by dragging and dropping the layers in the GIS layer control.

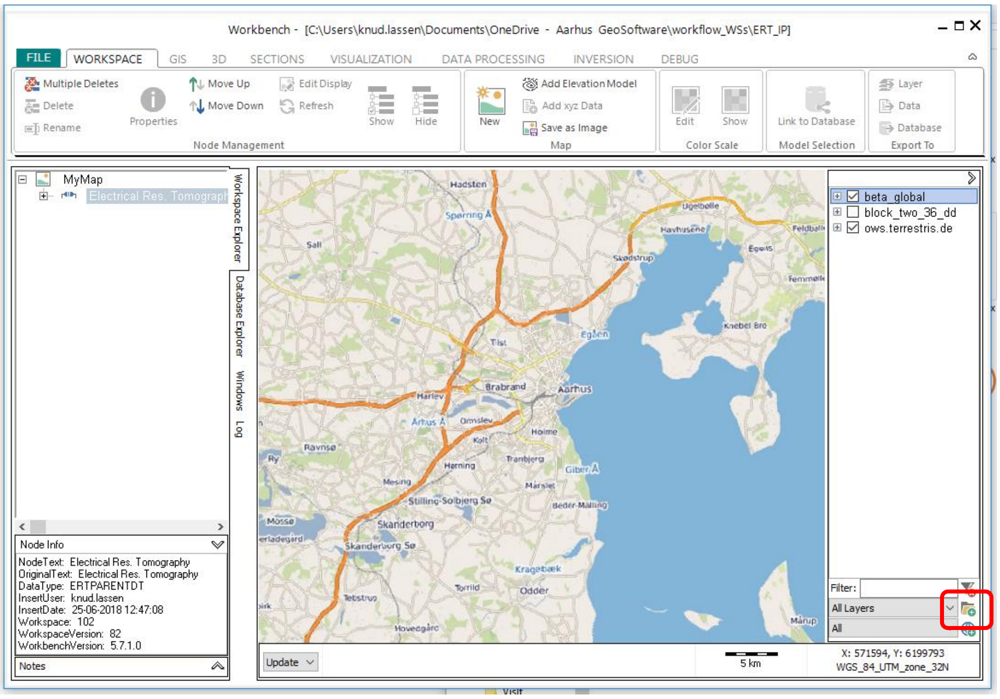


Figure 12

Online maps

Are maps that are accessible on a local server or a server on the internet using the WMS protocol. Using one of the default WMS servers is an easy way of getting a background map for the workspace, some countries and institutions also provide detailed topographical maps, aerial photos and other specialized map layers as WMS services, read more here: http://www.ags-cloud.dk/Wiki/WH_WMSLayers . A WMS layer is added by clicking the small globe below the button highlighted in Figure 12, a number of servers is suggested by default and more can be added.

Creating an offline background map from an inline map

It is always possible to save the current map view as an offline map e.g. so it can be used without internet connection. This is done in the following way:

1. Zoom to the wanted zoom level on the GIS map.
2. Go to workspace tab and press “Save as image”. A window will open and the image from the GIS map can be saved as a .tif file which is georeferenced. Remember if data or profiles are displayed on the GIS map, they will be saved with the image too.
3. Add the image to the GIS layer control by clicking the “open folder” button at the lower right corner and select the exported file.

Creating and using quality control themes

Model quality themes are a tool for assessing the quality of inversions and data processing by plotting different values on a color scale on the GIS maps.

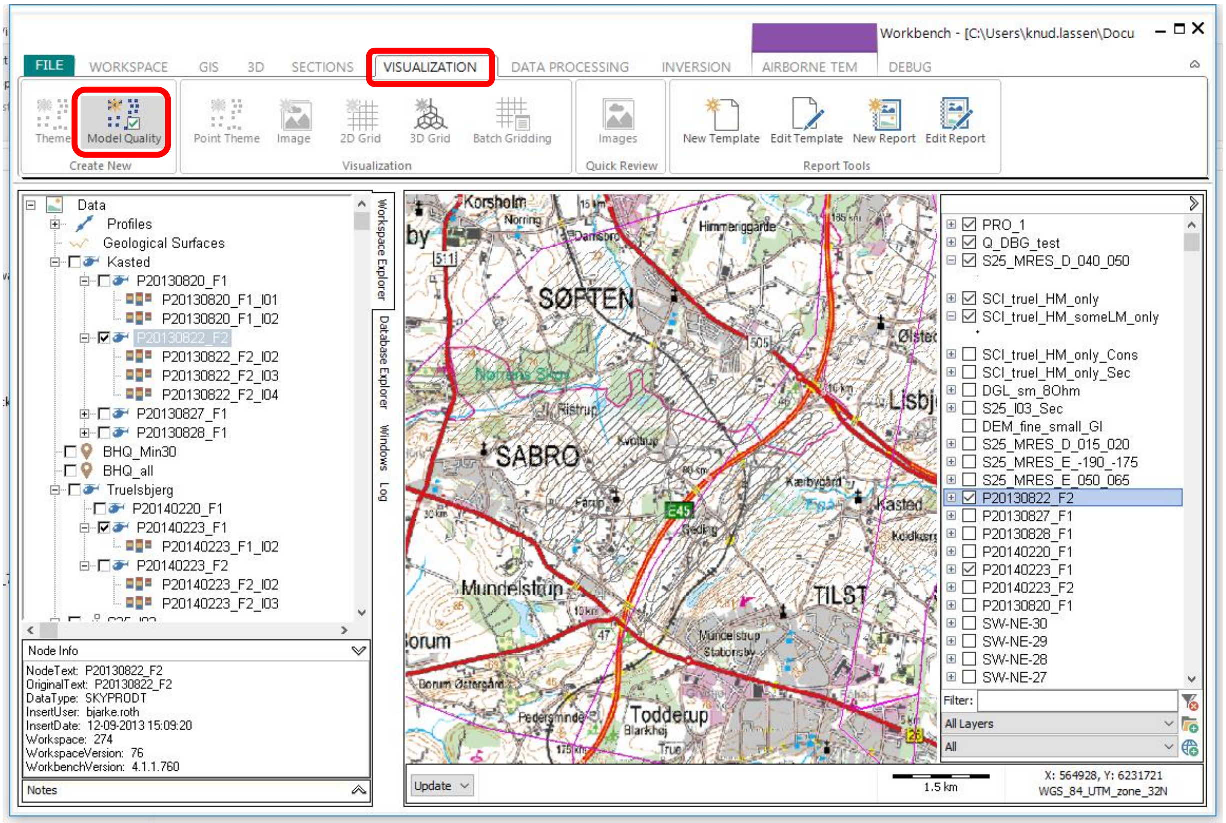
Common values to plot for the QC is total residual, data residual and dept of investigation (DOI).

The residuals can be used to determine how well the inverted models fit the data in different parts of the survey, bad data fit can indicate that the data are coupled, too noisy or that the inversion set-up isn't well suited for the geological setting, the data can then be reprocessed and/or reinverted in these areas, or it can be kept in mind that models from the areas can't be trusted for interpretation.

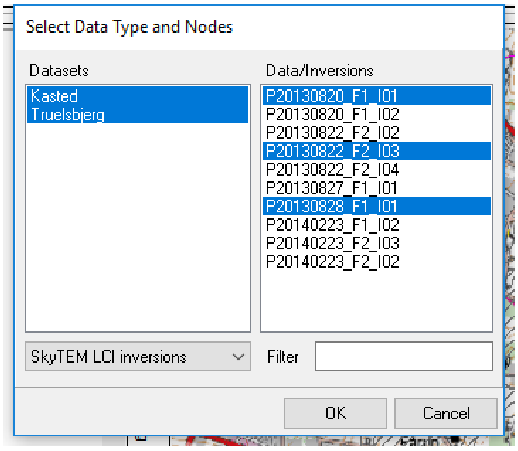
DOI can be used to determine whether the survey resolve the subsurface to a sufficient depth to serve the surveys purpose, and to guide the interpretation of data.

For airborne surveys it is also common to plot the instrument altitude and the difference between measured and inverted altitude. A high altitude e.g. when passing high trees or structure can explain a higher residual that otherwise can't be explained. The differences between measured and inverted altitude can serve as an indicator for either bad altitude processing or other problems in the inversion.

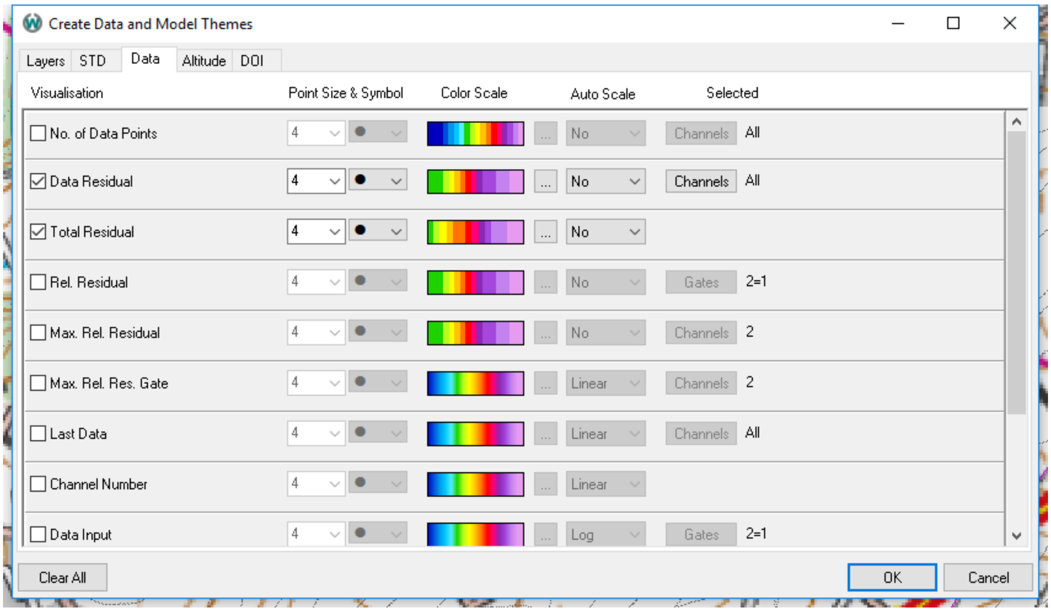
Model quality themes are created by navigating to the "Visualization" ribbon and selecting the "Model Quality" button:



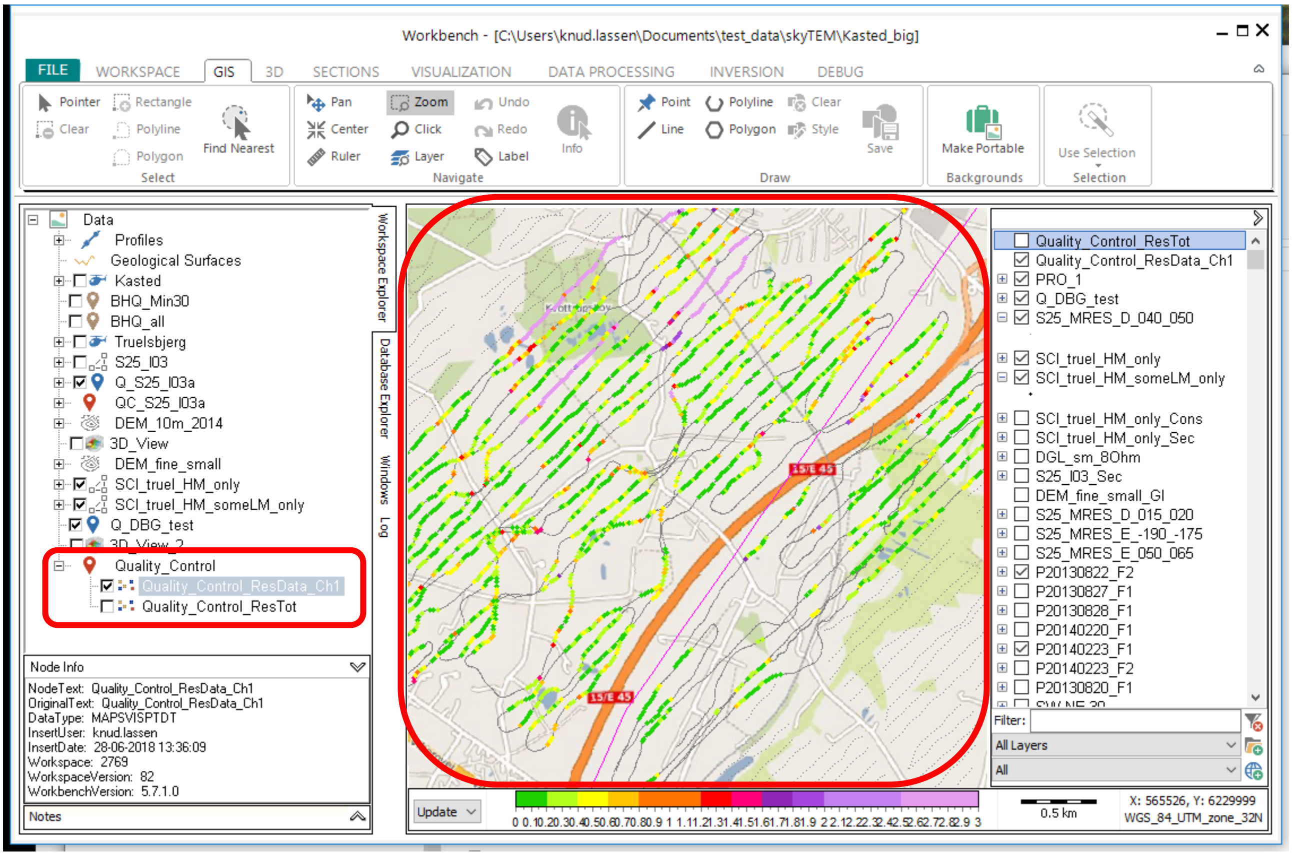
In the resulting dialogue box, it is possible to select which inversions from which datasets to work with. Note that more than one dataset and inversion can be selected by holding down “CTRL” while selecting:



After pressing “OK” the name of the quality control node must be entered and the values to be plotted selected, the point size, symbol and color scale can also be edited:



After pressing “OK” the node can be found in the workspace explorer under the entered name, and the different values can be selected to be plotted as colored symbol on the GIS map:

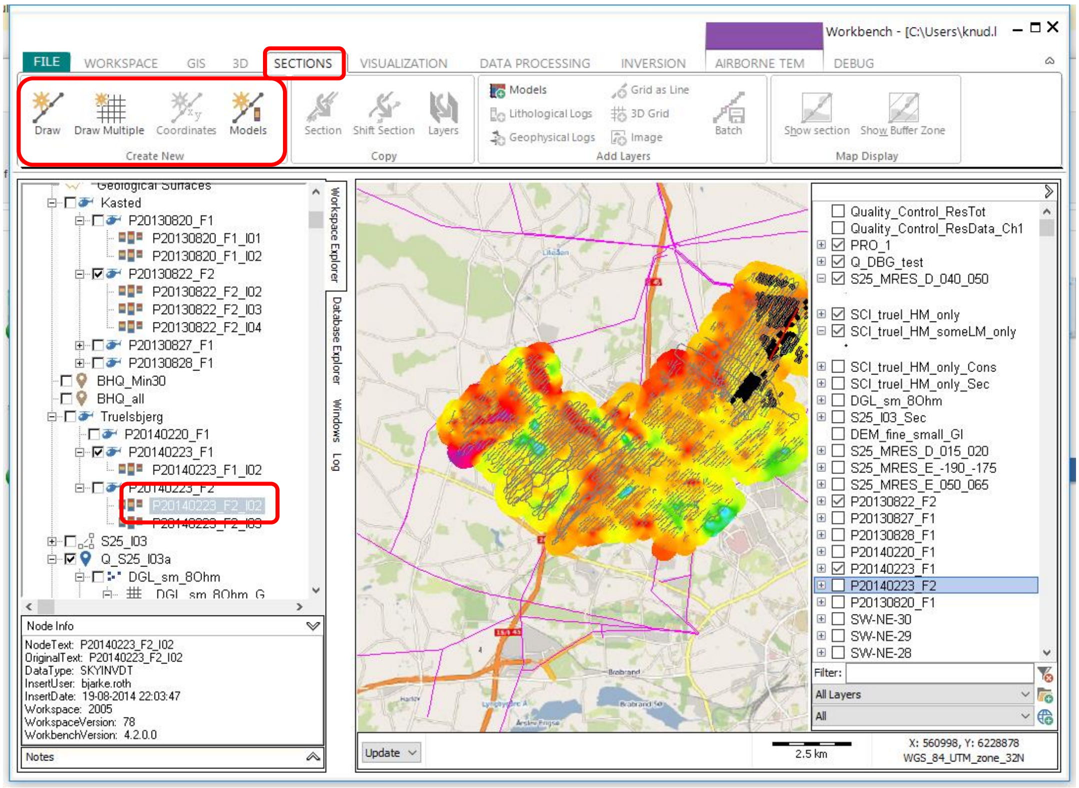

Workbench - [C:\Users\knud.lassen\Documents\test_data\skyTEM\Kasted_big]
FILE
WORKSPACE
GIS
3D
SECTIONS
VISUALIZATION
DATA PROCESSING
INVERSION
DEBUG
Pointer
Rectangle
Clear
Polyline
Polygon
Find Nearest
Select
Pan
Zoom
Undo
Center
Click
Redo
Ruler
Layer
Label
Info
Navigate
Point
Polyline
Clear
Line
Polygon
Style
Save
Draw
Make Portable
Backgrounds
Use Selection
Selection
Data
Profiles
Geological Surfaces
Kasted
BHQ_Min30
BHQ_all
Truelsbjerg
S25_I03
Q_S25_I03a
QC_S25_I03a
DEM_10m_2014
3D_View
DEM_fine_small
SCI_truel_HM_only
SCI_truel_HM_someLM_only
Q_DBG_test
3D_View_2
Quality_Control
Quality_Control_ResData_Ch1
Quality_Control_ResTot
Workspace Explorer
Database Explorer
Windows
Log
Node Info
NodeText: Quality_Control_ResData_Ch1
OriginalText: Quality_Control_ResData_Ch1
DataType: MAPSVISPTDT
InsertUser: knud.lassen
InsertDate: 28-06-2018 13:36:09
Workspace: 2769
WorkspaceVersion: 82
WorkbenchVersion: 5.7.1.0
Notes
15/E 45
15/E 45
Quality_Control_ResTot
Quality_Control_ResData_Ch1
PRO_1
Q_DBG_test
S25_MRES_D_040_050
SCI_truel_HM_only
SCI_truel_HM_someLM_only
SCI_truel_HM_only_Cons
SCI_truel_HM_only_Sec
DGL_sm_8Ohm
S25_I03_Sec
DEM_fine_small_GI
S25_MRES_D_015_020
S25_MRES_E_-190_-175
S25_MRES_E_050_065
P20130822_F2
P20130827_F1
P20130828_F1
P20140220_F1
P20140223_F1
P20140223_F2
P20130820_F1
Filter:
All Layers
All
Update
0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1 1.1 1.2 1.3 1.4 1.5 1.6 1.7 1.8 1.9 2 2.1 2.2 2.3 2.4 2.5 2.6 2.7 2.8 2.9 3
0.5 km
X: 565526, Y: 6229999
WGS_84_UTM_zone_32N

Create sections and add models and data to these

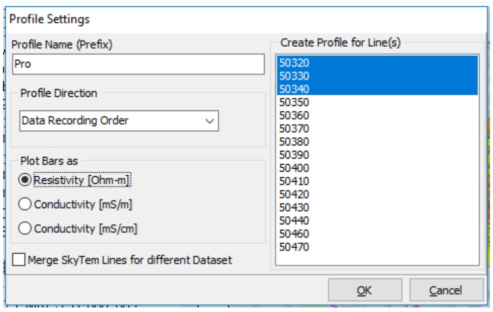
Sections are the classical way to display inversion results from different inversions together with borehole data, logs, geological surfaces etc.

Sections can be created in two ways:

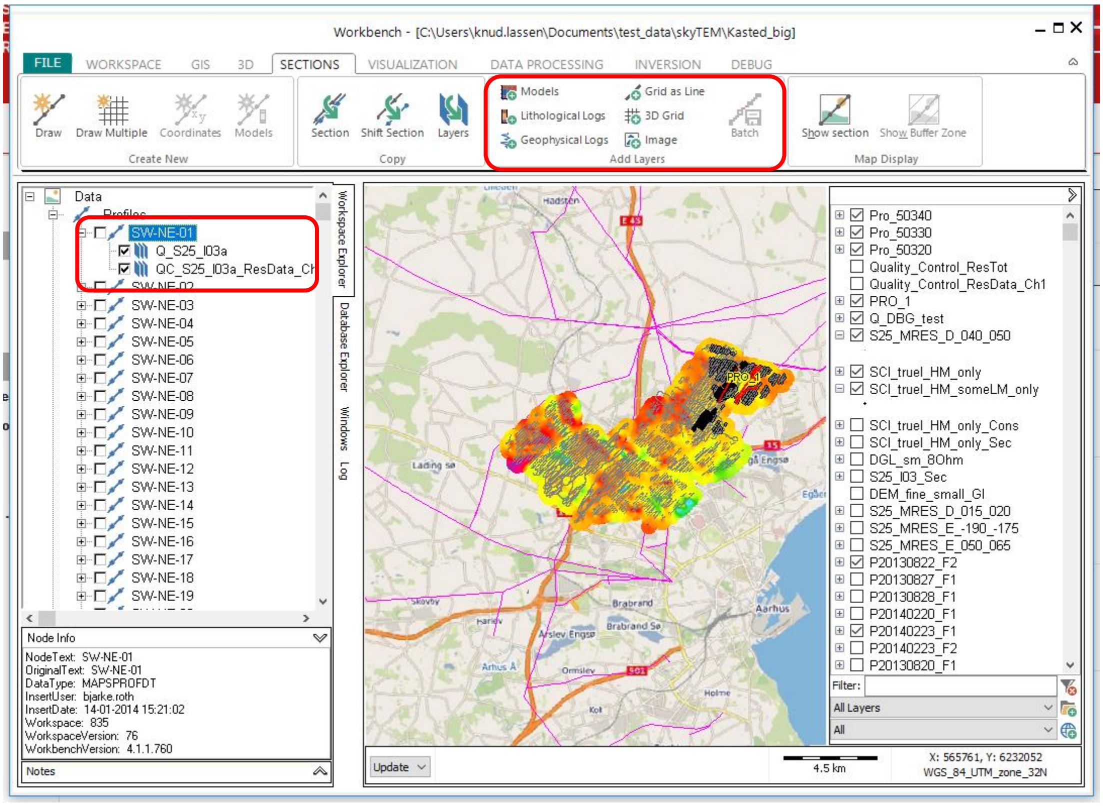
1. Under the "Sections" ribbon in the "Create new" group "Draw" can be selected, and the section can then be drawn in the GIS, single click for each bend in the section and double click to end the drawing and. After double clicking you will be prompted to name the section.
2. Draw section from inversion. A section can also be drawn directly from inversion results, in this way the section will lie exactly on top of data, and the inverted models can be added to the section when it is created. To create a section in this way, navigate to the "Sections" ribbon, highlight the inversion node in the workspace explorer by clicking it once, and select "Models" in the "Create new" box:



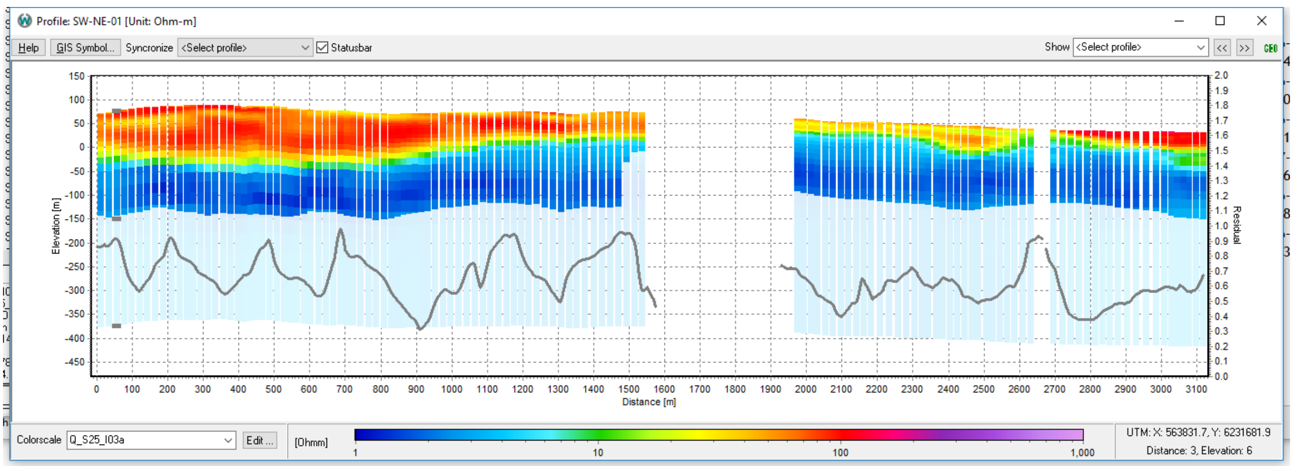
If the inversion node contains more than one data line you will be prompted to select for which lines you want to create sections, you can also select the name and how to plot the data at the section:



After pressing OK you will be prompted to select the color scale to plot the models on, and to name the layer on the profile. Once created the sections can be found under the profiles node, each profile can contain several layers, different layers can e.g. be inverted resistivity and IP parameters, geophysical logs, borehole information, quality control parameters like data residual (grid the theme and use grid as line to get it on the profile). The layers can be toggled on and off, when a layer is toggled off it will not be shown on the profile, a profile is opened for display by ticking the check mark next to the profile. New layers can be added to the profile by highlighting the profile in the workspace explorer and adding layers in the "Add layers" group. Remember that the different kinds of data must be loaded into the workspace before they can be added.



Finally, an example of a section of inverted SkyTEM data with the data residual plotted together with the models:



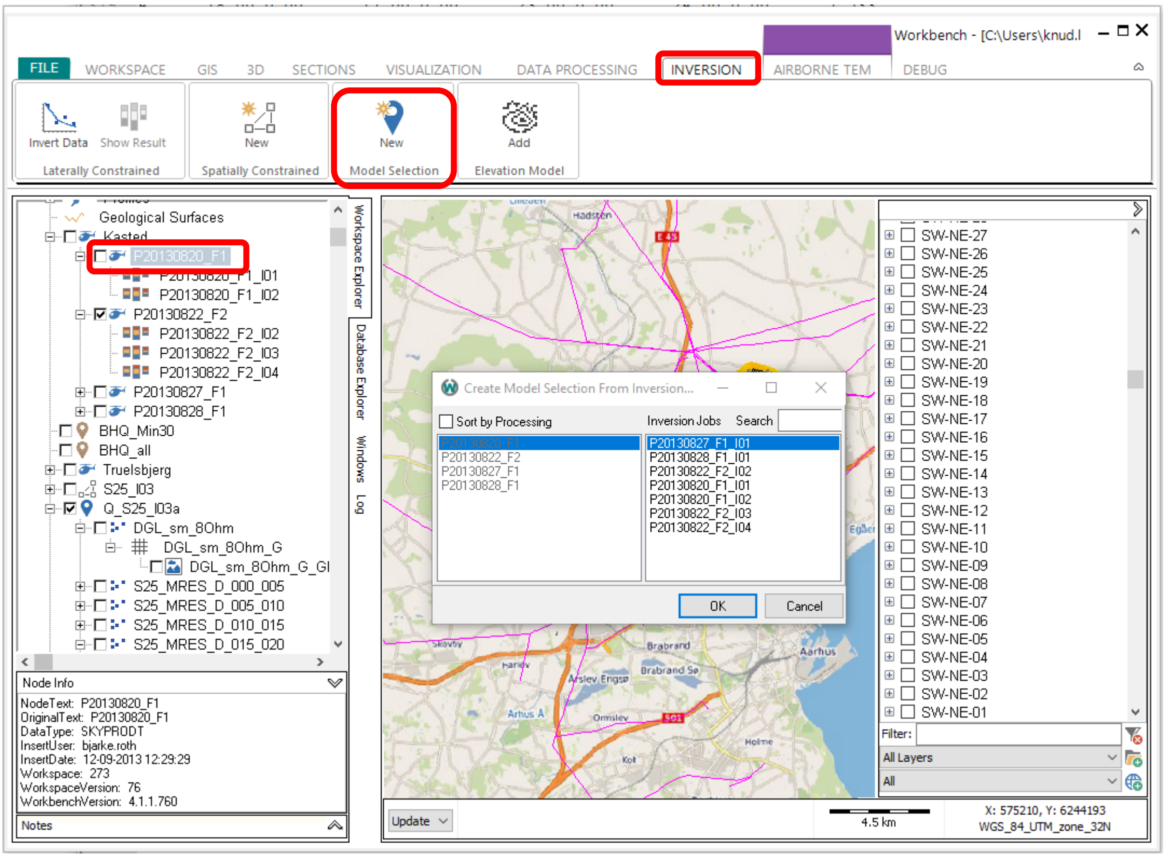
Note that the range and type (logarithmic or linear) of the axis can be edited by double clicking the axis.

Creating and using themes

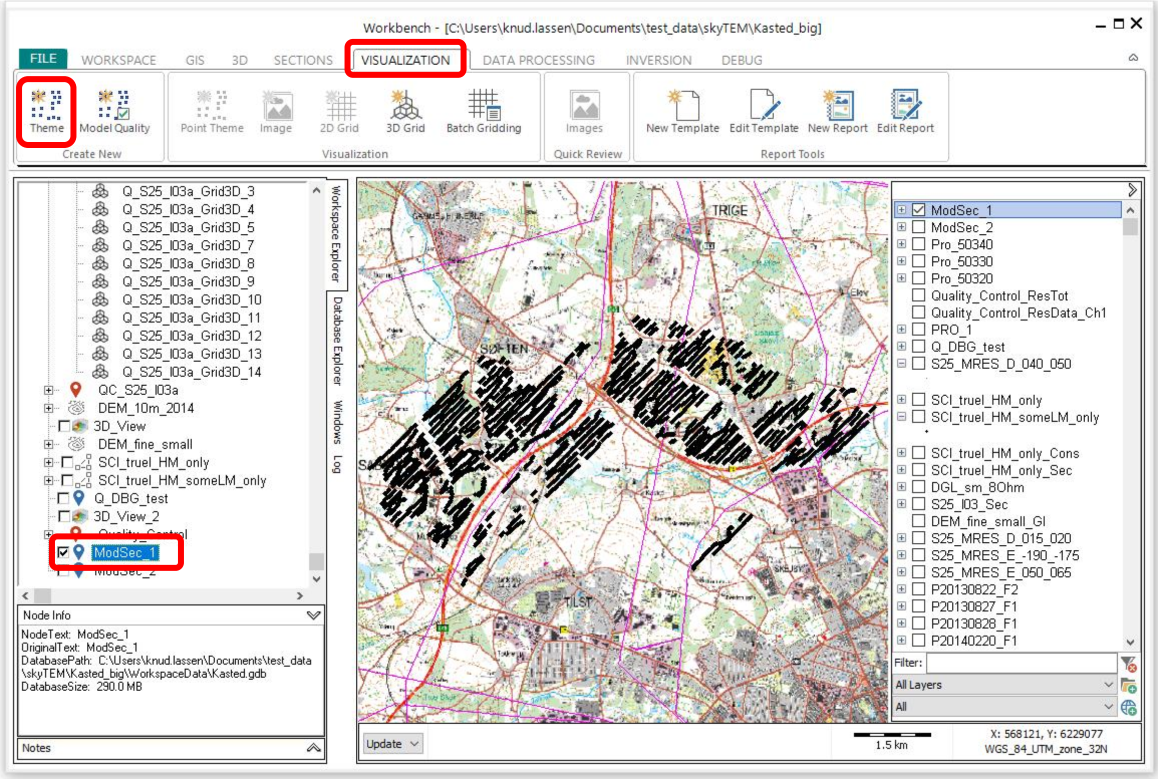
Themes are the tool for visualizing results on the map e.g. the mean resistivity of a given depth interval, the depth to a good conductor, the resistivity of the third layer in the model, or thickness of a body with a given resistivity.

A theme contains these values in the discrete points at which the models are located, they can be visualized either as colored icons at these points (See: Point themes) or as surface covering interpolated grids (See: 2D grids).

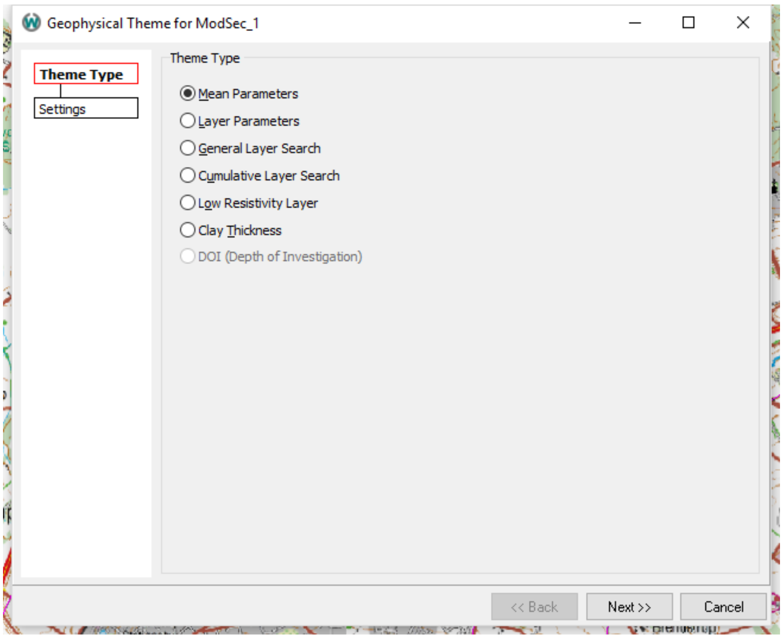
A theme is created from a model selection. A model selection is created from the "Inversion" ribbon by highlighting a processing or inversion node and selecting "New" from the "Model Selection" group. A model selection can contain models from inversions from different processing nodes by removing the ticker from "Sort by processing" after selecting the desired inversions and pressing "ok" the name of the model selection must be entered. Once the selection is created it can be found in the workspace explorer.



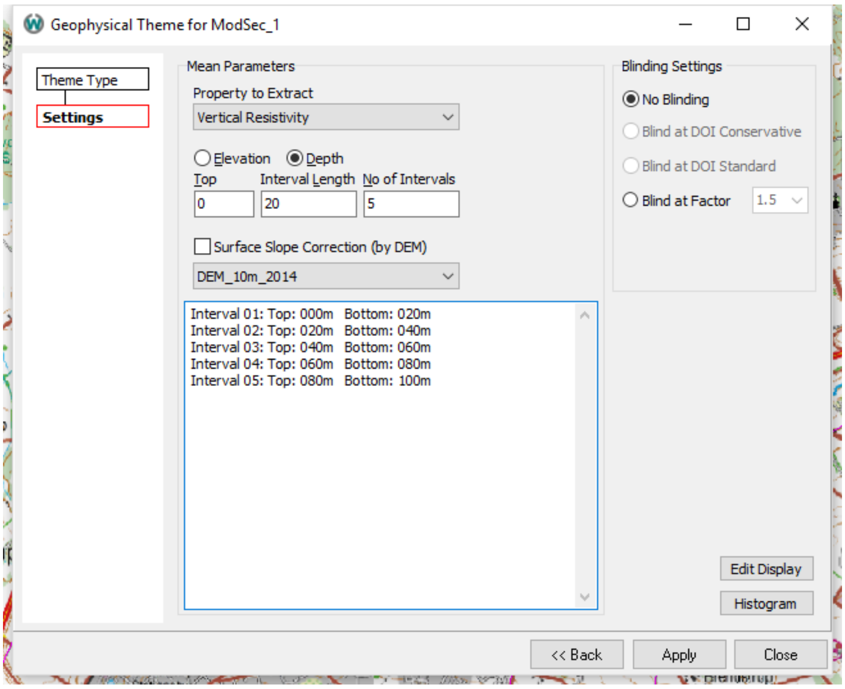
After creating the model selection, the actual theme can be created by highlighting the model selection in the workspace explorer and selecting "Theme" in the "Visualization" ribbon:



The following dialog box will then appear:



From this list the type of theme must be selected, the simplest theme is the “Layer Parameters” theme, this theme simply contains the value of the selected layer at the different positions, we will continue by creating a mean resistivity theme by selecting “Mean Parameters”. For a comprehensive walkthrough of the different types of themes use the F1 help from this window. In the next menu the property to be extracted, the number of intervals and the thickness of the layers must be specified:

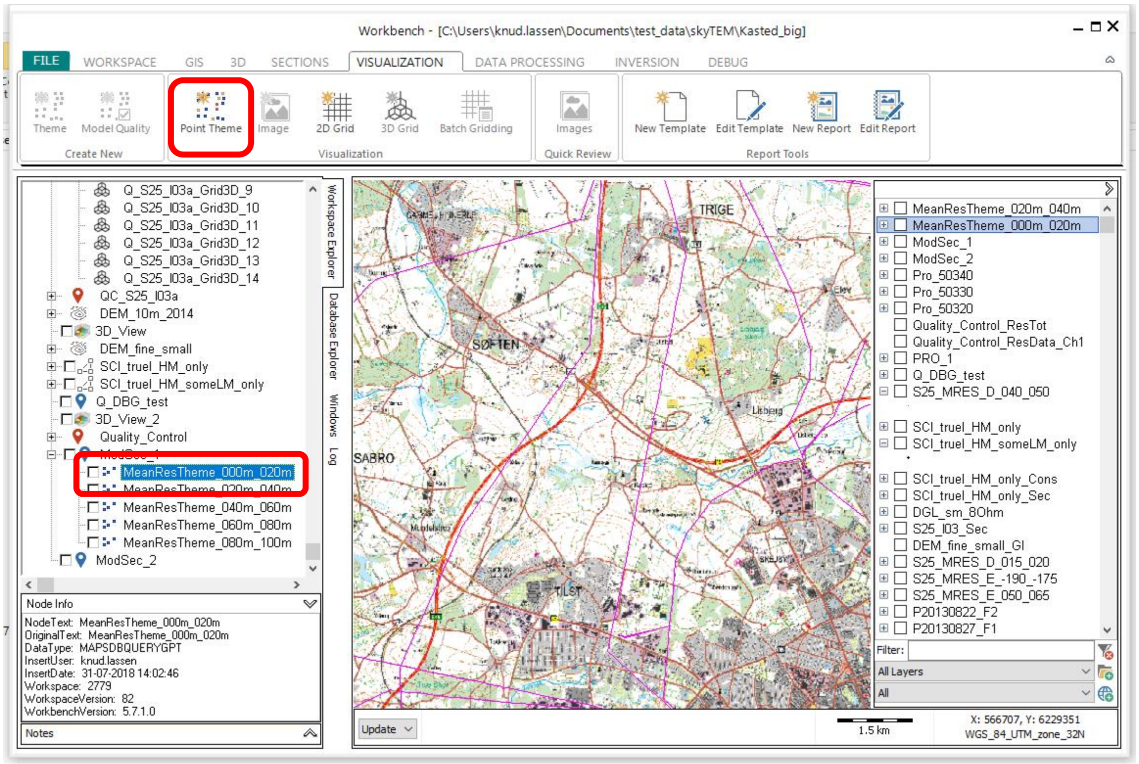


The themes are created by selecting "Apply". The themes can now be visualized as points or 2D grids.

Visualizing themes

Points

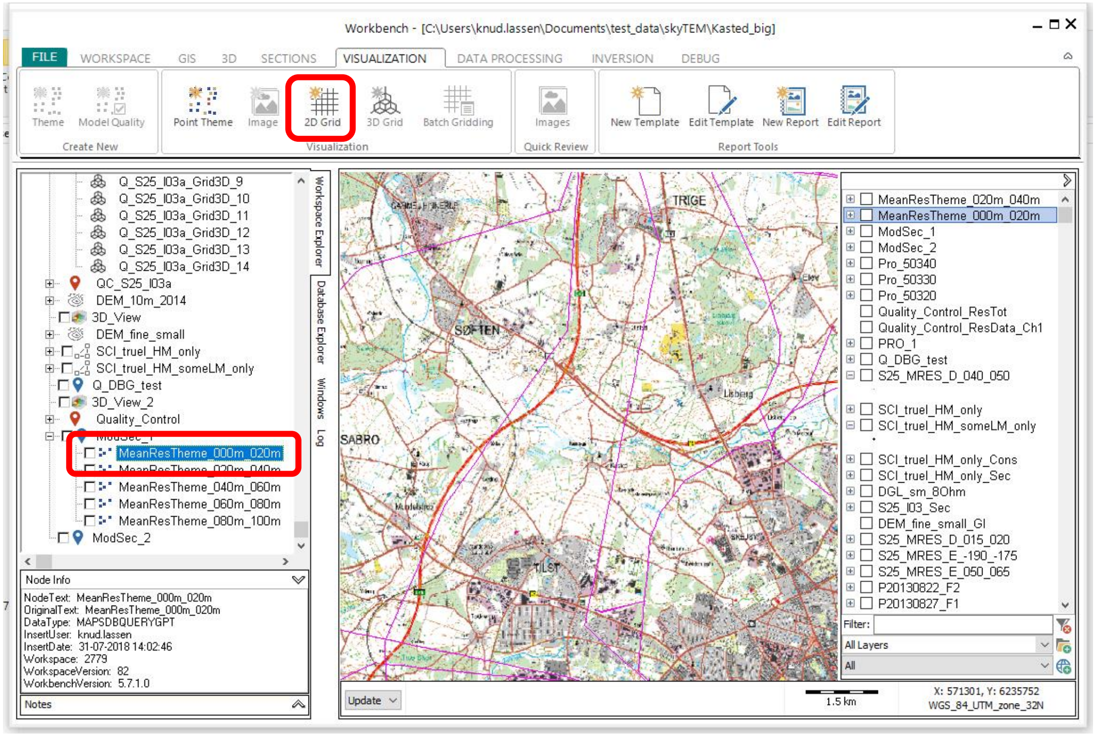
To plot the gridded values as colored points in the GIS window highlight the theme and press "Point Theme"



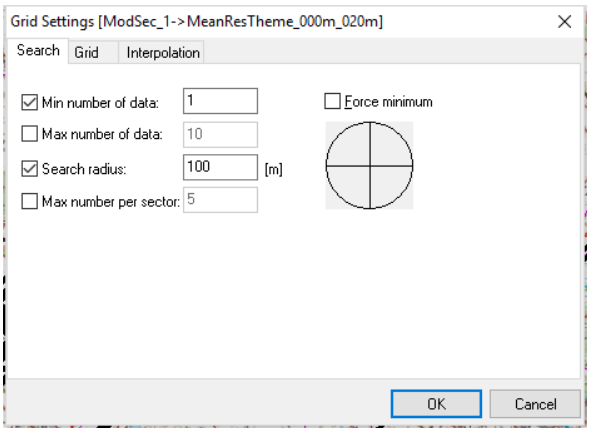
Select the color scale, point size and point shape in the dialog box and press “OK” and select the name of the theme, the point theme can now be found in the workspace explorer.

2D Grids

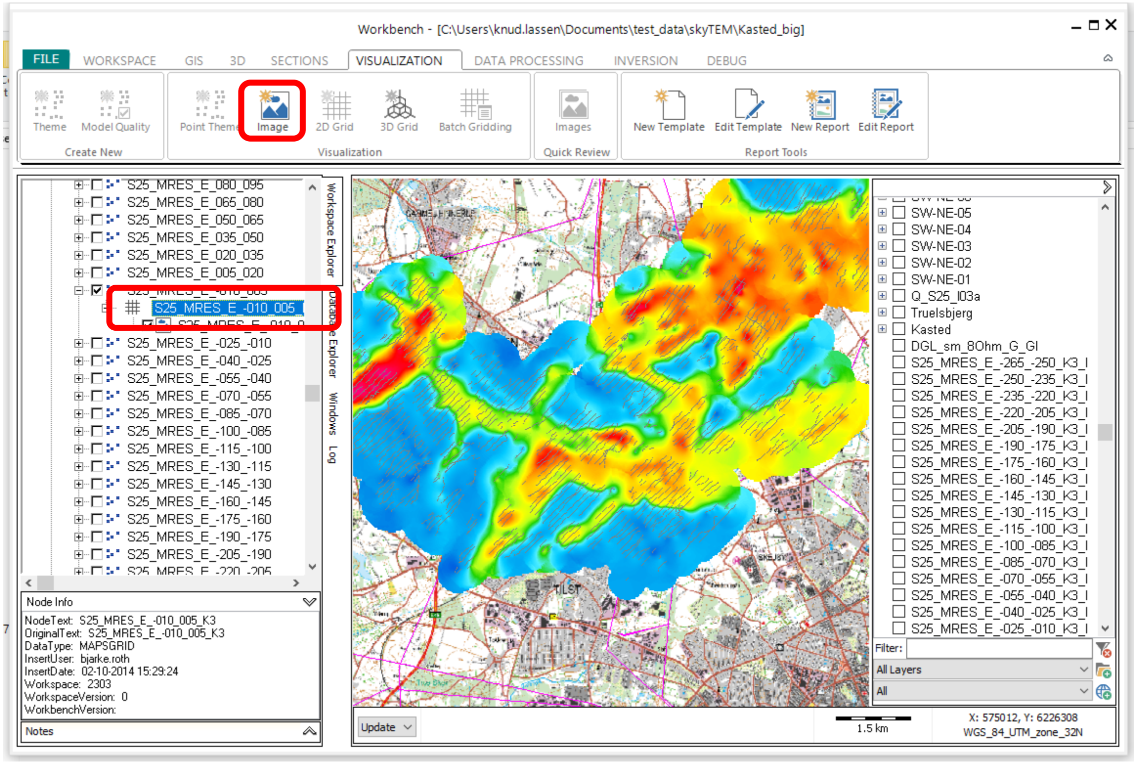
The other way of presenting themes are as surface covering interpolated grids, these are created by highlighting the theme and pressing “2D Grid”



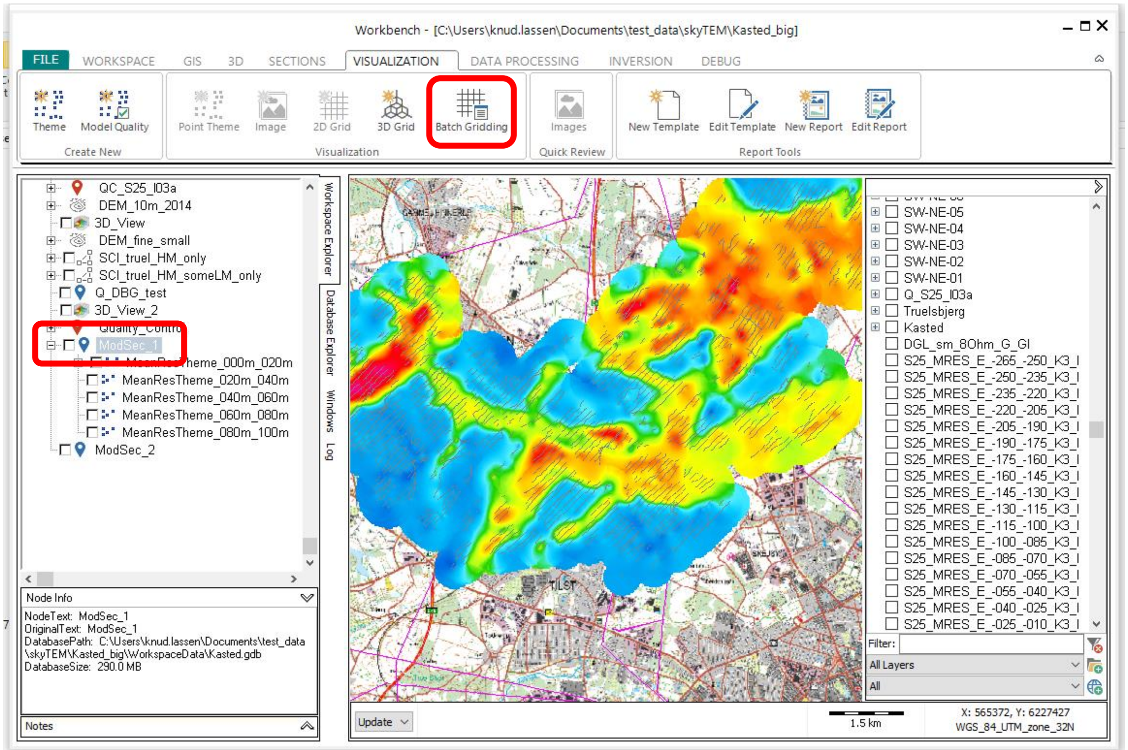
The following menu box will appear:



In this the search radius, grid spacing, and interpolation routine must be selected, refer to the F1 help for further specification, after pressing “OK” the grid must be named, after the grid is calculated an image must be created to display the grid. The first image is automatically created after calculating the grid, the color scale and name for the image must be selected and the image can then be found in the workspace explorer. Subsequent images e.g. with different color scales can be created by highlighting the grid and selecting “Image” in the visualization ribbon:



It is also possible to make 2D grids and images for several themes in a model selection in one go by highlighting the model selection and selecting “Batch Gridding”:

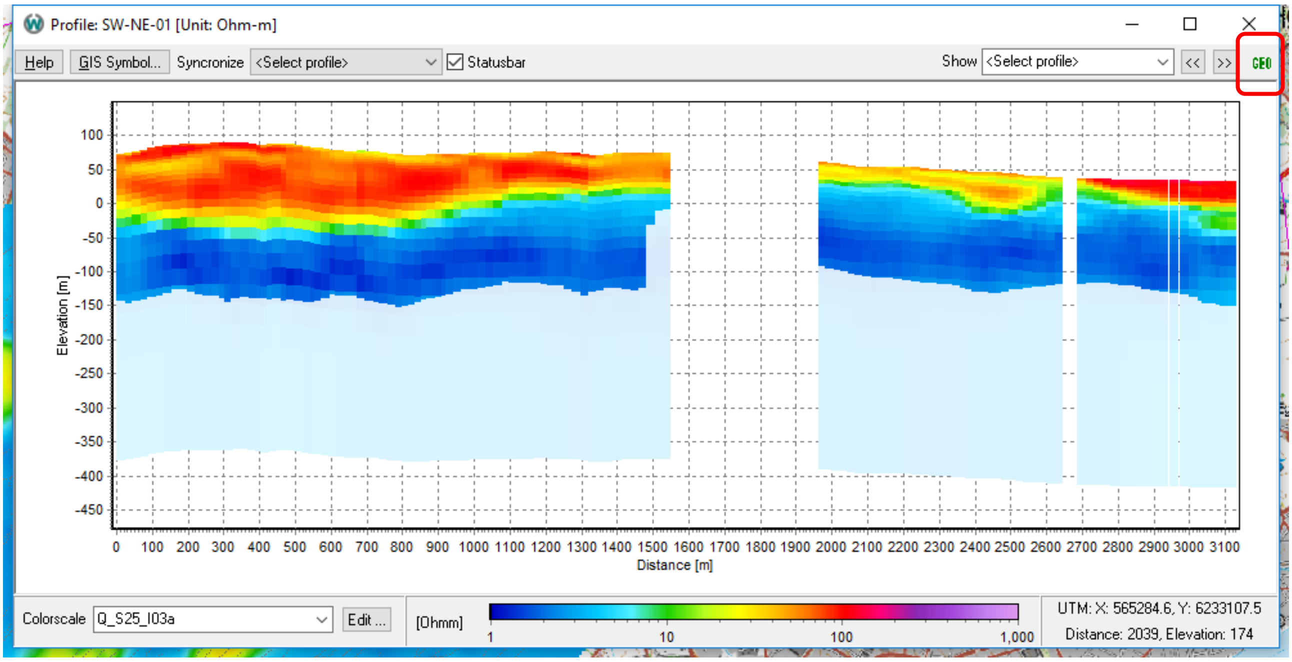


Drawing and gridding geological surfaces

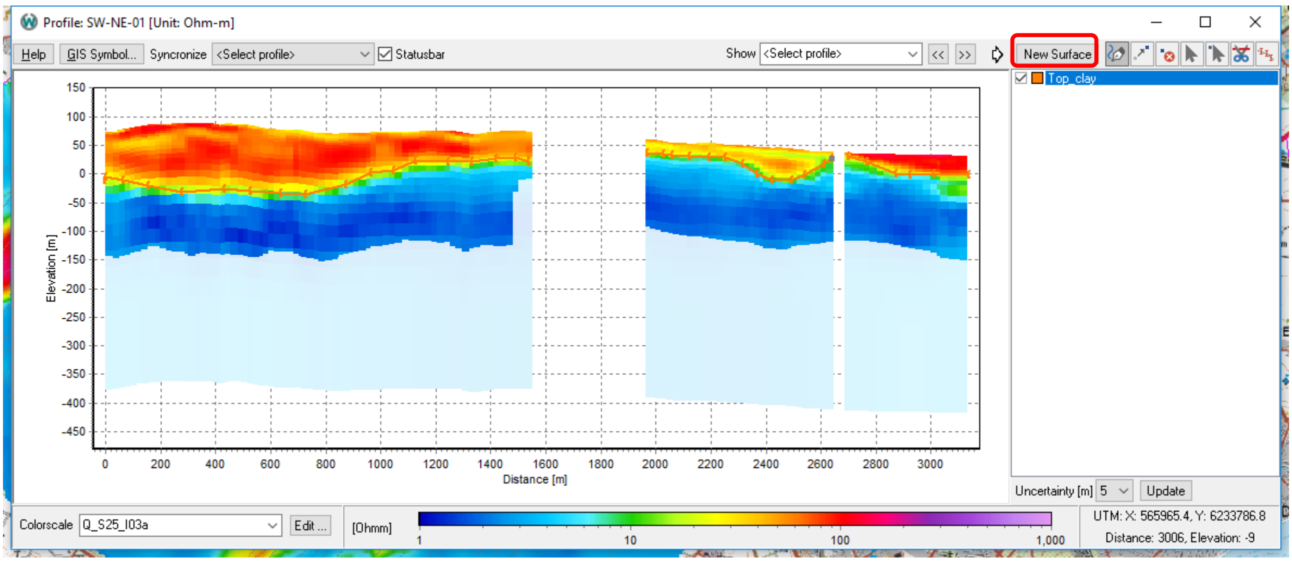
Geological surfaces are the interpretation tool of Aarhus Workbench, these can be used for tracking geological bodies and other targets, such as the interface between sand and clay, the top and bottom of a buried landfill or the water table in an aquifer, across different sections and data types.

Once drawn the surfaces can be gridded to interpolate between the sections so that maps of e.g. the water table or the thickness of a pollution can be created. The surfaces can also be exported as shape or xyz files.

A geological surface is created by pressing the “GEO” bottom at the top right in an open profile:



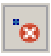
This opens a new menu next to the profile:



A new surface is created by pressing “New Surface” and entering a name for the surface. The surface can be edited using the controls seen below:


Draw on profile, select this tool to start drawing the surface on the profile. The profile is drawn by clicking on the profile.

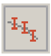

Move point, this tool makes it possible to drag and drop misplaced points


Delete points, deletes the selected point, in drawing mode the last drawn point will be deleted, to delete other points select the point using the “select points” tool.


Default select tool – only used to select other things than geological surfaces.


Select points


Break surface, this tool breaks the surface at the point selected using the “select points” tool.


Show/hide error bars.

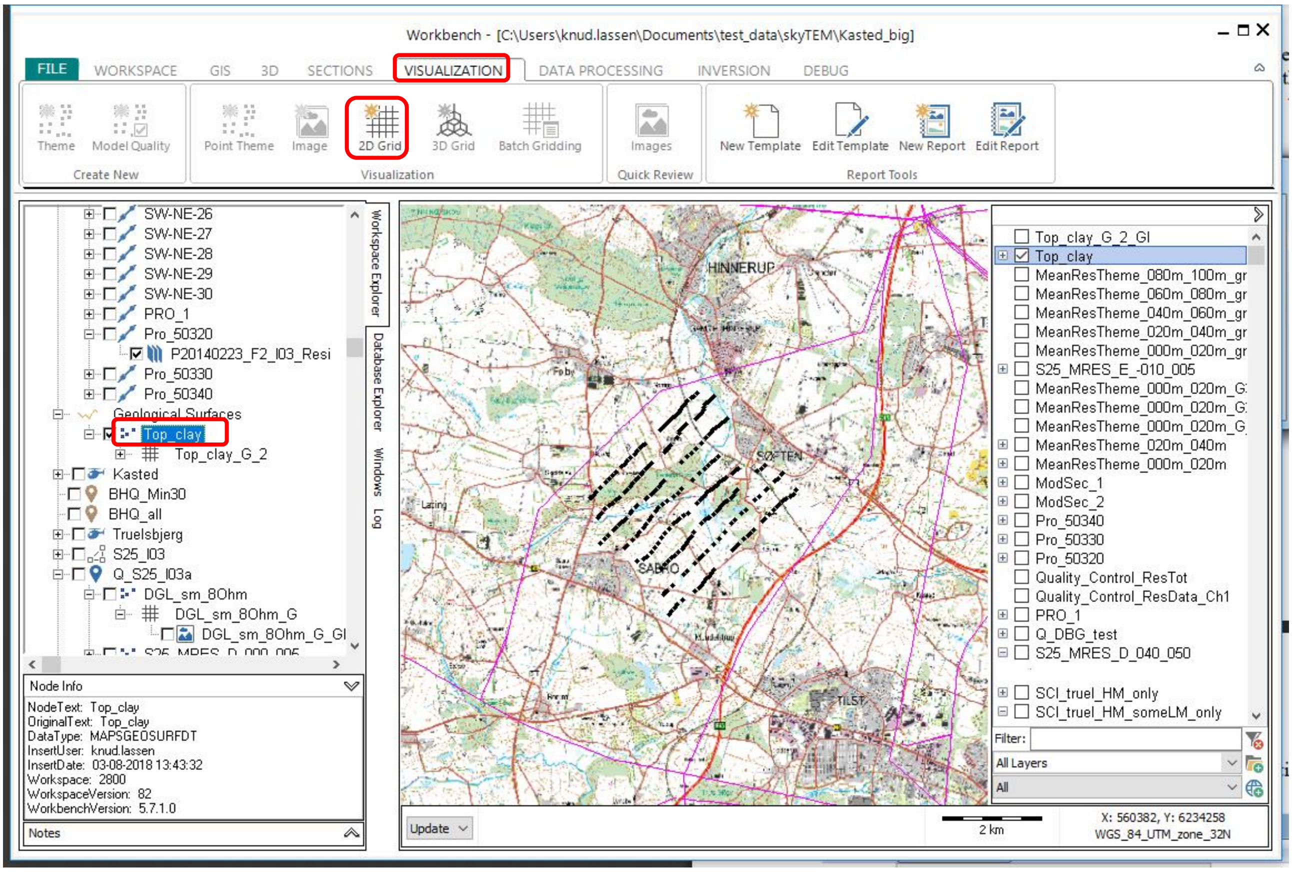
Change uncertainty, changes the uncertainty on the selected points when pressing “update” and on any new points drawn, this feature makes it possible to express the uncertainty on the exact location of the geological surface.

Colors of surfaces can be changed either by clicking the colored box next to the surface name or right clicking at a surface and choosing “update color”.

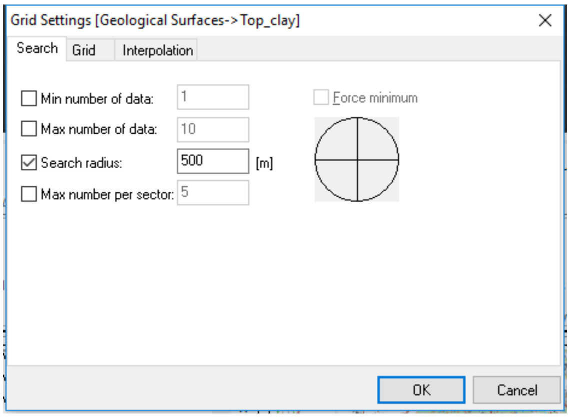
The location of the points on the geological surface can be shown on the GIS map by checking the name of the geological surface in the workspace explorer under “geological surfaces”. This can be useful in getting an overview over which surfaces are drawn for which profiles. Before this is possible it is necessary to right-click the name of the geological surface in an open profile and press “Update GIS points”.

Gridding of geological surfaces

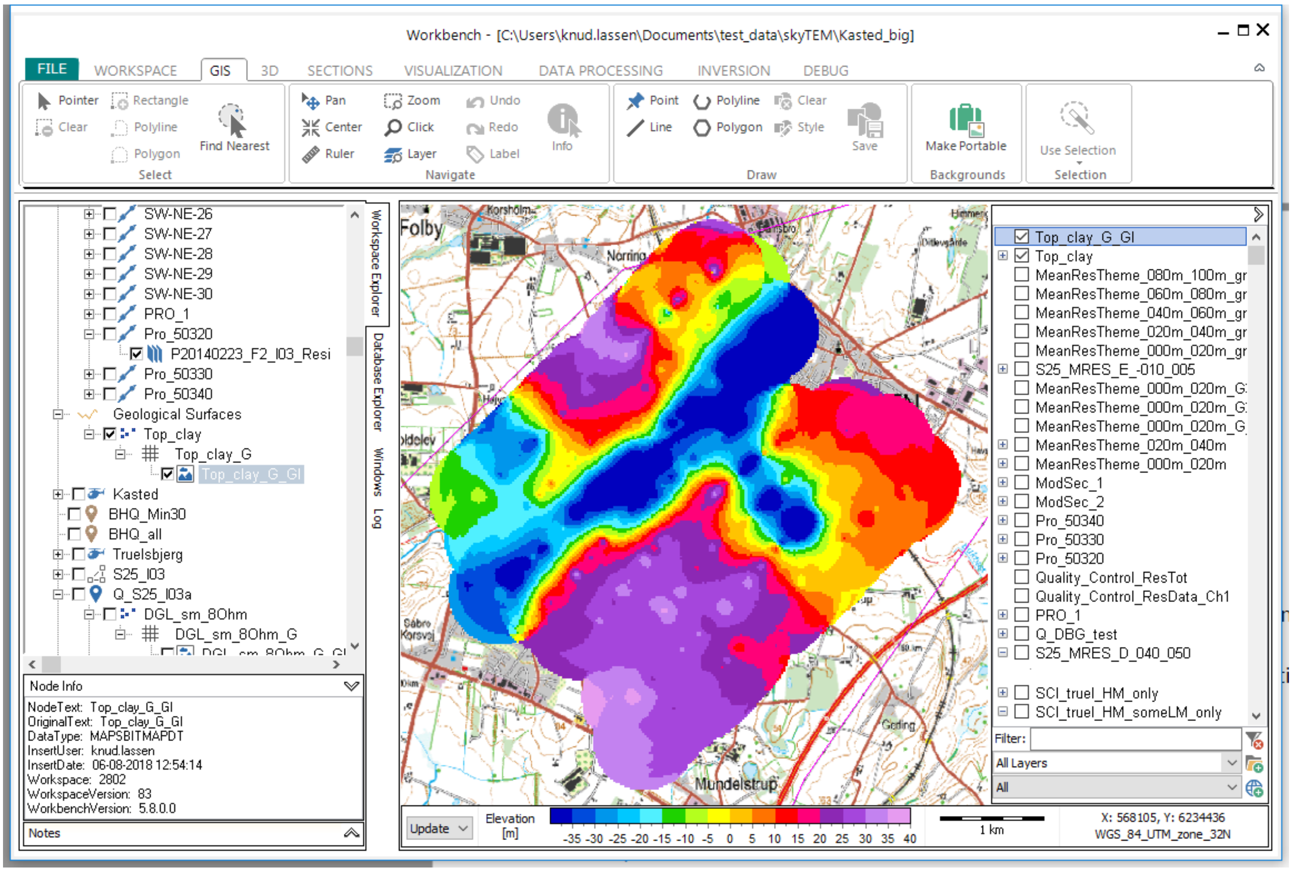
Once the geological surface is drawn on all sections/profiles in the area of interest the surface can be gridded to be shown on a horizontal map. The grid is created by navigating to the “VISUALIZATION” ribbon, highlighting the geological surface to grid in the workspace explorer and pressing “2D Grid”.



You will be presented with the following dialog box in which you can specify different parameters for the gridding, for a detailed description of the different parameters please refer to the F1 help. When pressing "ok" the new grid can be named.



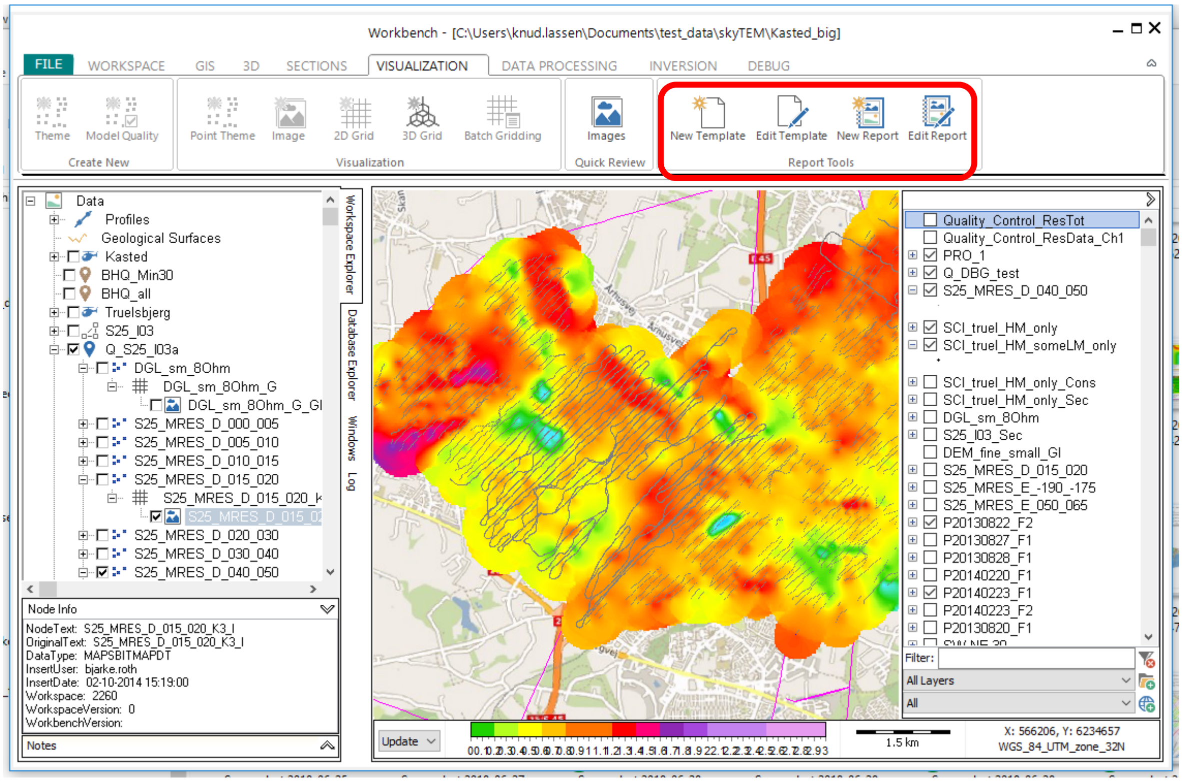
Once the grid is created it can be found in the workspace explorer under the geological surface, to display the grid on a map an image must be created for the grid. This is done by highlighting the grid in the workspace explorer and selecting "Image" under in the "VISUALIZATION" ribbon, you will then be prompted to select the color scale for the image and to provide a name for the image. Below is seen an example of such an image displaying the elevation of the top of the clay layer for a small part of a Danish survey:


Workbench - [C:\Users\knud.lassen\Documents\test_data\skyTEM\Kasted_big]
FILE
WORKSPACE
GIS
3D
SECTIONS
VISUALIZATION
DATA PROCESSING
INVERSION
DEBUG
Pointer
Rectangle
Clear
Polyline
Polygon
Find Nearest
Select
Pan
Zoom
Undo
Center
Click
Redo
Ruler
Layer
Label
Info
Navigate
Point
Polyline
Clear
Line
Polygon
Style
Save
Draw
Make Portable
Backgrounds
Use Selection
Selection
SW-NE-26
SW-NE-27
SW-NE-28
SW-NE-29
SW-NE-30
PRO_1
Pro_50320
P20140223_F2_I03_Resi
Pro_50330
Pro_50340
Geological Surfaces
Top_clay
Top_clay_G
Top_clay_G_GI
Kasted
BHQ_Min30
BHQ_all
Truelsbjerg
S25_I03
Q_S25_I03a
DGL_sm_8Ohm
DGL_sm_8Ohm_G
Workspace Explorer
Database Explorer
Windows
Log
Node Info
NodeText: Top_clay_G_GI
OriginalText: Top_clay_G_GI
DataType: MAPSBITMAPDT
InsertUser: knud.lassen
InsertDate: 06-08-2018 12:54:14
Workspace: 2802
WorkspaceVersion: 83
WorkbenchVersion: 5.8.0.0
Notes
Folby
Norring
Mundelstrup
Top_clay_G_GI
Top_clay
MeanResTheme_080m_100m_gr
MeanResTheme_060m_080m_gr
MeanResTheme_040m_060m_gr
MeanResTheme_020m_040m_gr
MeanResTheme_000m_020m_gr
S25_MRES_E_-010_005
MeanResTheme_020m_040m
MeanResTheme_000m_020m
ModSec_1
ModSec_2
Pro_50340
Pro_50330
Pro_50320
Quality_Control_ResTot
Quality_Control_ResData_Ch1
PRO_1
Q_DBG_test
S25_MRES_D_040_050
SCI_truel_HM_only
SCI_truel_HM_someLM_only
Filter:
All Layers
All
Update
Elevation
[m]
-35 -30 -25 -20 -15 -10 -5 0 5 10 15 20 25 30 35 40
1 km
X: 568105, Y: 6234436
WGS_84_UTM_zone_32N

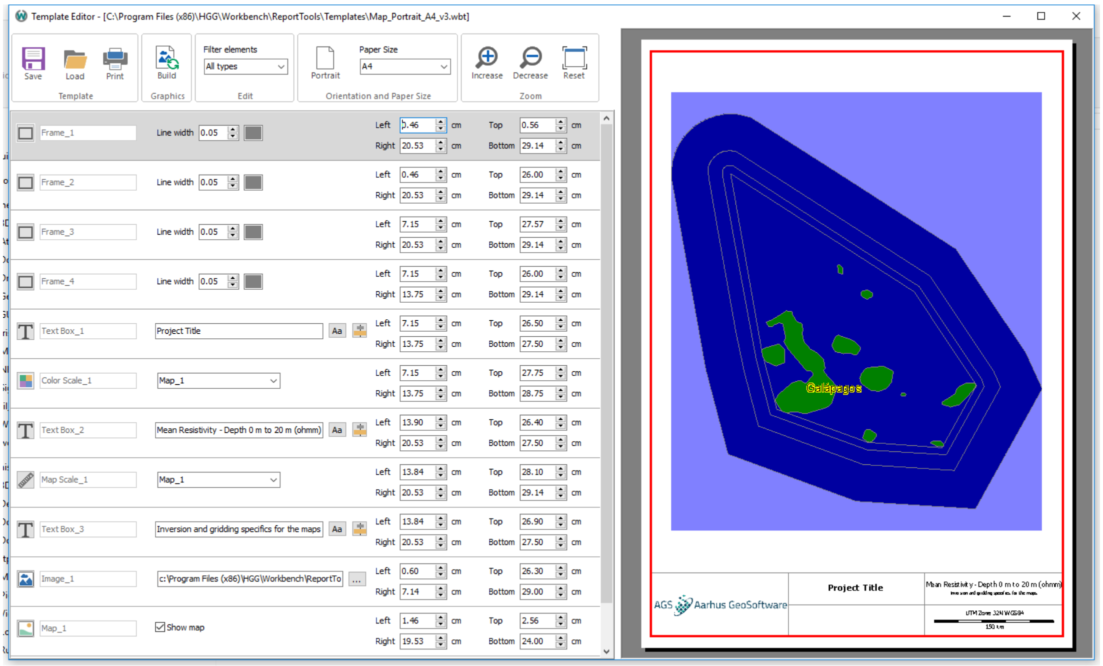
Creating a PDF report from template

The report tool is a tool to create high resolution, professional looking reports from the different data found in the workspace, the reports can include maps, profiles, themes etc. The reports are generated from predefined templates that defines the layout of the report e.g. paper format, location of logos, overview maps, profiles and other objects. Once a template is created it is easy to recreate reports of the same type for surveys at different locations, or to e.g. make a daily report from a long field campaign.

Templates and reports can be created and edited from the Visualization ribbon in the report tools box:

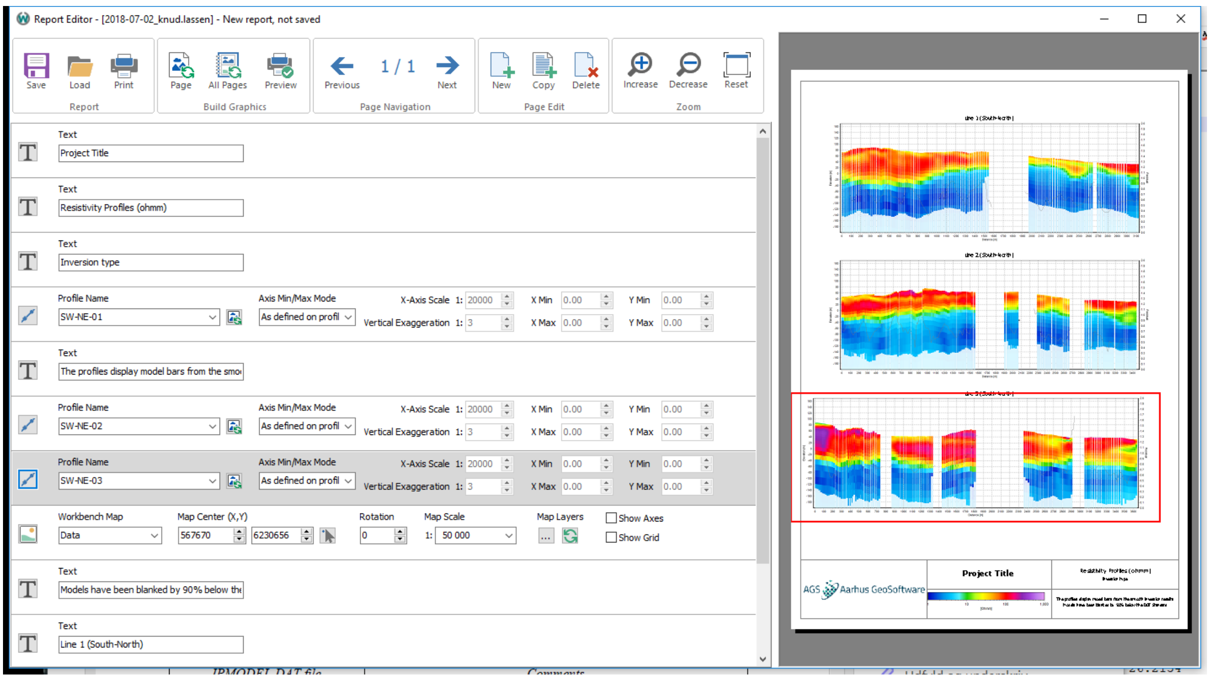


Each report page is built from the following elements: frames, text boxes, maps, legends, map scales, north arrows, images, profiles and color scales. By selecting either new template or edit template the template editor is opened:



To the left the editor is shown, to the right a preview is displayed. It is possible to edit the size, location and appearance of each element. New elements are added by selecting the icon to the left of an element and selecting “Copy item”. It is also possible to change the type of the element or to delete elements in this way. The preview is updated by pressing “Build.” For further explanation of the different options use the “F1” help. To use the template to create a report, save the template and select “New Report”.

The report editor is very similar to the template editor, but instead of editing the type and location of the object it is now possible to edit the content e.g. which profiles are displayed and the text in the textboxes. Changes take effect once “page” of “all pages” is selected in the “Build graphics” menu. It is also possible to add or remove pages. A report can be saved and edited later. To create a PDF of the report, select “print” and print the report to a PDF printer. Use the “F1” help for explanation of the individual features.



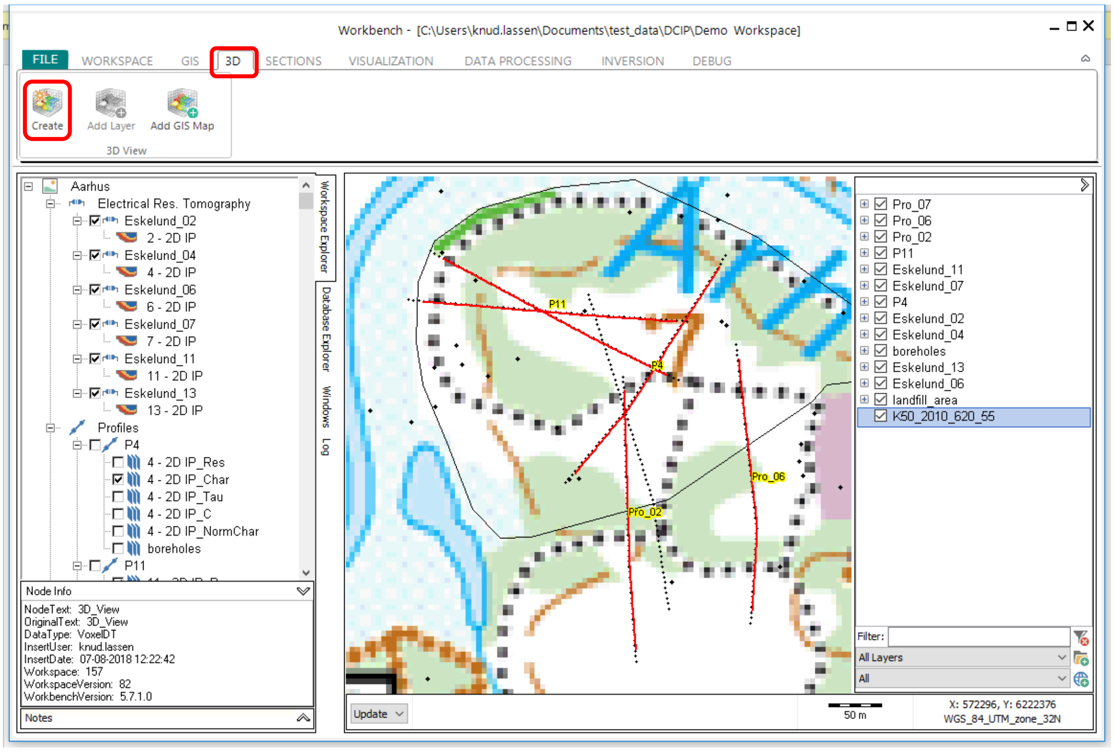
Using the 3D viewer

The 3D viewer is a powerful tool for presenting data and for gaining a deeper understanding of the inversion results by being able to rotate and view them from all directions.

The 3D viewer can display profiles, maps, 2D and 3D grids and boreholes from the workspace as well as external grids and images loaded as .vtk, .tiff, .xyz, .grd, .dat, and .bgr files.

Creating 3D view and adding profiles from the workspace explorer

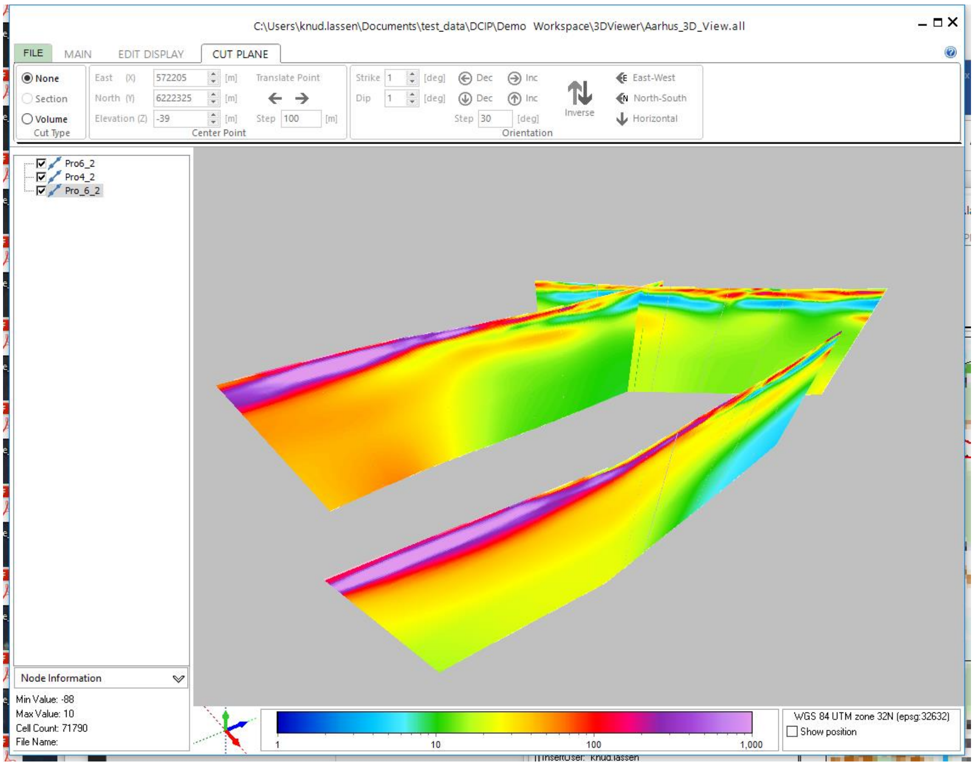
To create a new 3D view, navigate to the “3D” ribbon and select “Create”:



After entering a name for the 3D view it will open in a separate window. To add a profile to the 3D view, highlight the element in the workspace explorer and add it by pressing “Add layer” in the “3D” ribbon. Once loaded the different profiles can be toggled on and off in the menu to the left (see below).

Note that the profile will be added in the form that will be displayed when opening the profile, e.g. the data types that are checked under the profile when it is added to the 3D viewer, and the desired color scale must also be chosen at the profile prior to adding the profile to the 3D viewer. Finally; when adding models to the profile the “show interpolated bars” option must be selected to obtain a continuous resistivity section instead of discrete columns of resistivities.

The 3D viewer with 3 profiles loaded in:



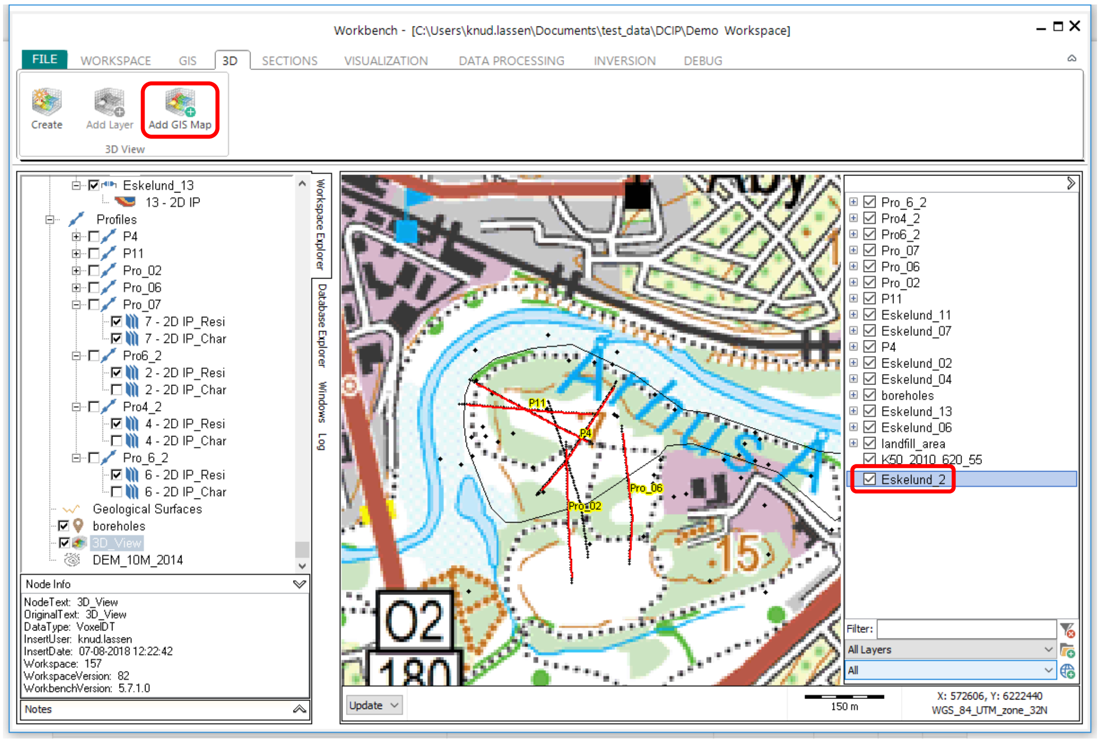
Navigating in the 3D viewer

The basic tool for navigating in the 3D viewer is the mouse, the view can be rotated by holding down the left mouse button and moving the mouse, zooming in and out can be done using the scroll wheel, the content off the viewer can be panned or moved relative to the center of rotation by holding down the right mouse button and dragging the content, the center of rotation can be shown by checking the “center” group in the “main” ribbon, this can make it easier to position the model just right.

Note that the “right mouse button panning” always takes place in the plane parallel to the screen, so to move the content in another plane/direction it is necessary rotate the view using the left mouse button.

Adding maps, grids and images to the 3D viewer

All maps and pictures present in the workspace can be added to the 3D view, there are two main types of pictures usually found in the workbench, the first are maps and aerial photos, these are found in the GIS layer control to the right in the workbench and can be added by highlighting the layer and selecting “Add GIS map” in the “3D” tab:



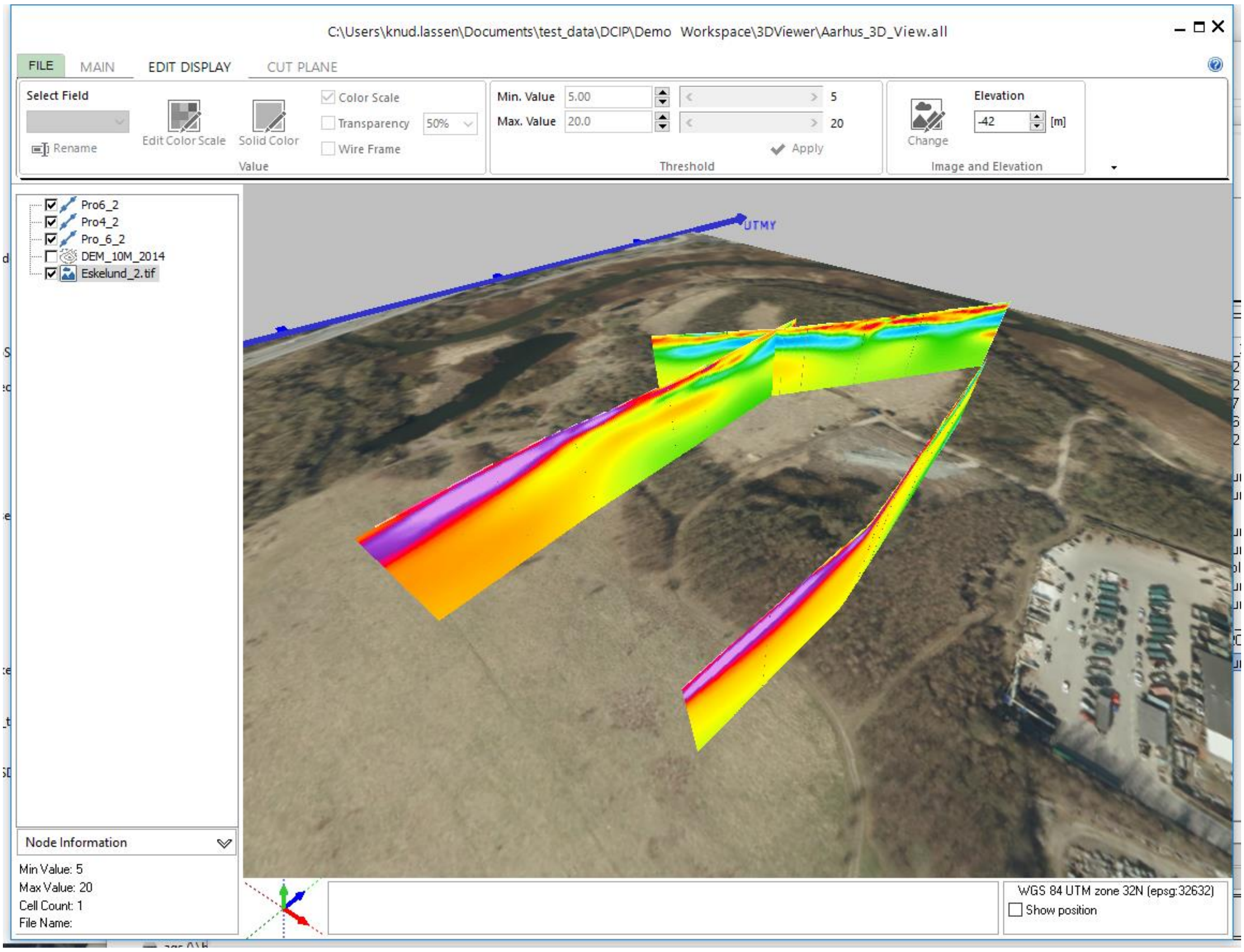
Location referenced images can also be imported directly to the 3D viewer by pressing “File->Load file”.

The other type of images are those created from gridded geophysical themes or gridded geological surfaces, these can be highlighted in the workspace explorer and added to the 3D view in the same ways as sections.

For both types of images, it is possibly to import them either as flat images or draped on a grid. If imported as a flat image it is possible to control the elevation of the image relative to the other elements in the viewer under the “Edit display” ribbon in the “Image and elevation” group. To drape the image on a grid this must be imported first, this is done by highlighting the grid in the workspace explorer and selecting “Add Layer” in the same way as with a section. The grid will usually be a DEM (digital elevation model) when importing maps and aerial photos, and the grid from which the image was generated for themes and geological surfaces.

Once a grid is present in the workspace a popup will ask whether to import the image as a flat image or draped on the grid when an image is imported:

An example of the 3 profiles from before displayed on top of an aerial photo:

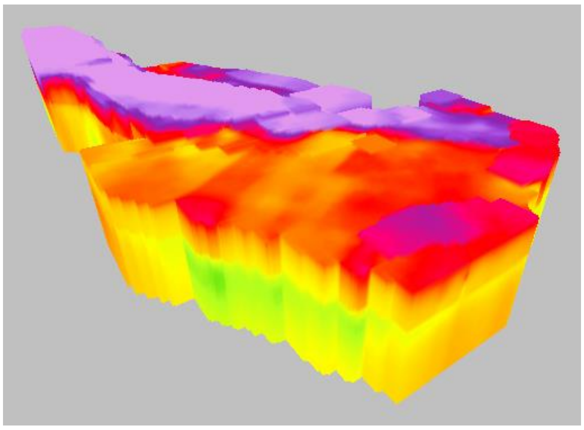


Cut planes, thresholding and 3D grids

Cut planes and thresholding are features that are mainly relevant for data that are gridded in 3D.

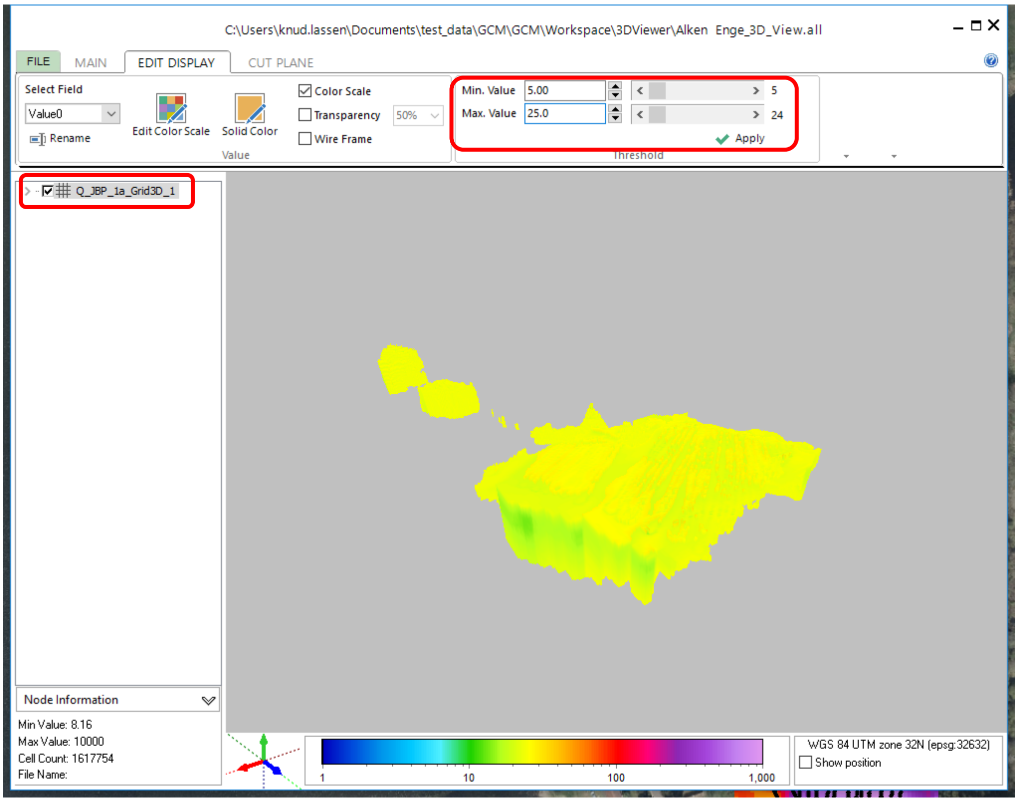
3D grids are added as all other data types by highlighting in the workspace manager and selecting "Add Layer", they can also be loaded in directly as VTK files e.g. from external inversion or modeling software e.g. Res3DInv.

The most intuitive way to demonstrate these functions are through examples. Below a 3D grid based on GCM data has been imported into the 3D viewer without further changes:

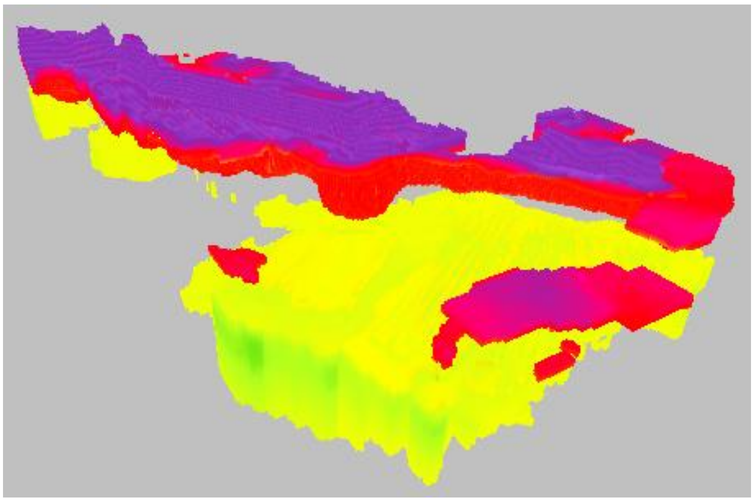


Thresholding

Thresholding makes it possible to only show the parts of the grid with certain resistivity values e.g. between 5 and 25 Ωm. It is done by highlighting the relevant object in the menu to the left in 3D viewer, navigation to the “EDIT DISPLAY” ribbon, selecting the desired min and max value and clicking “Apply”:

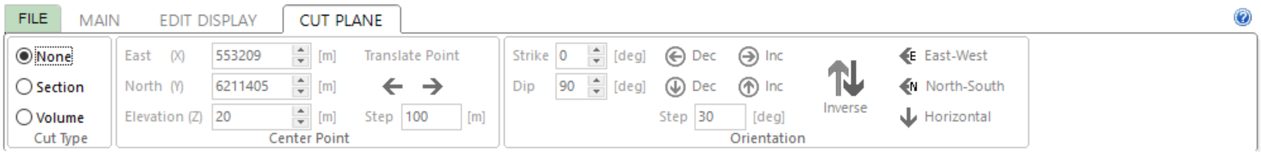


It is possible to show more than a single range of resistivities at a time, this is done by adding subnodes to the object. This is done by highlighting the object and selecting “Add Subnode” in the “Main” ribbon. Each of these subnodes can then be given different threshold values as shown below:

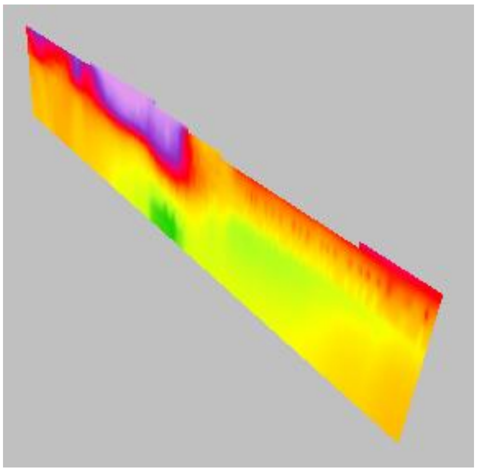


Cut planes

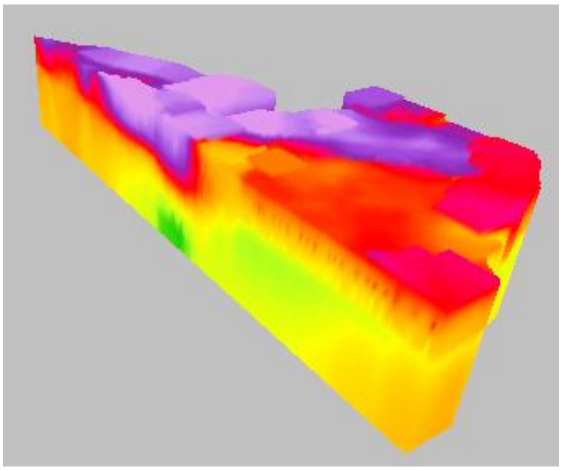
Cut planes makes it possible to make cuts through data to show relevant regions, they are found on the “CUT PLANE” ribbon:



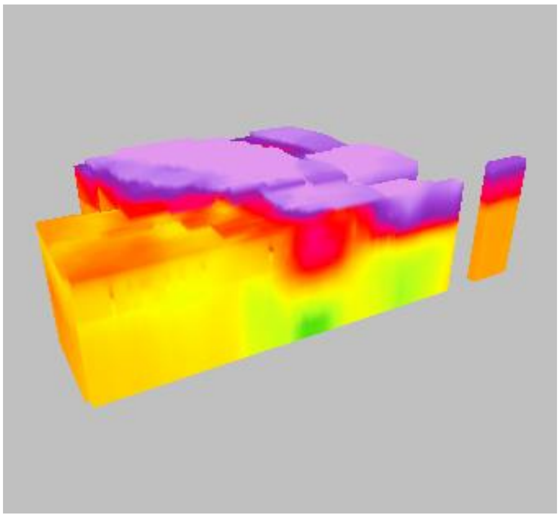
The default setting is “None”, there are two additional settings, namely “Section”, showing a slice through data:

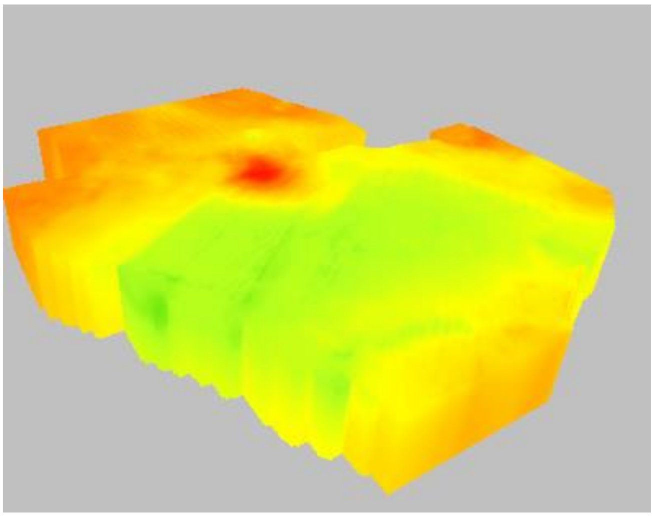


And “Volume” that cuts out part of the data:

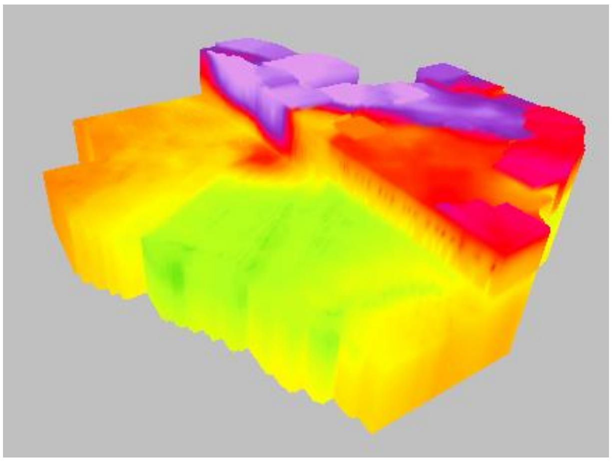


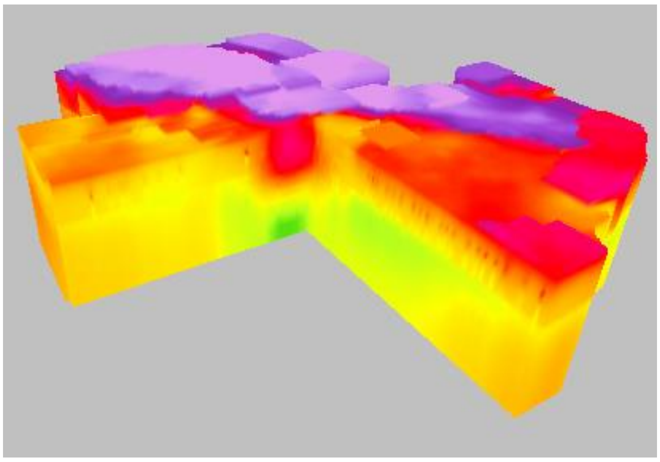
It is possible to change the location and orientation of the cut plane using the controls in the “Center Point” and “Orientation” groups:





As with thresholding is it possible to combine cut planes by using subnodes:





And as with all other tool in the workbench, use the F1 help to get a detailed explanation of the workings of every individual bottom.

Export

The pictures created with the 3D viewer can be saved by pressing “save image” in the “main” ribbon or copied directly to the clipboard by pressing “clipboard”. It is also possible to record the rotation of the 3D view into a little movie (Only possible from ver. 5.8), this is done by pressing “Rec Start” and naming the video file when prompted, all movements will be recorded until “Rec stop” is pressed.

Import of borehole information

This section describes how to import .bor and .rkt files in Aarhus Workbench, and how to load the borehole information and the corresponding rock type descriptions. Format specifications for the different formats can be found at: http://www.ags-cloud.dk/Wiki/W_GuidesBoreholes

1) Go to "File", press "Import" and select "Lithological logs" in the Boreholes tab (**Error! Reference source not found.**). Either you can add the boreholes to an existing database or you can create a new database. Press OK.

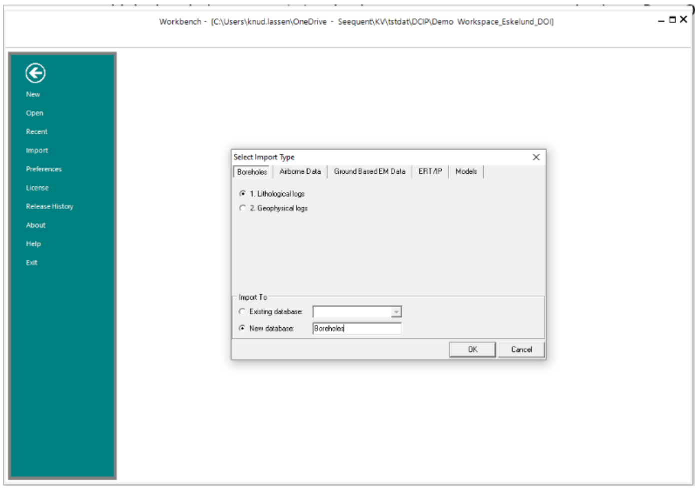
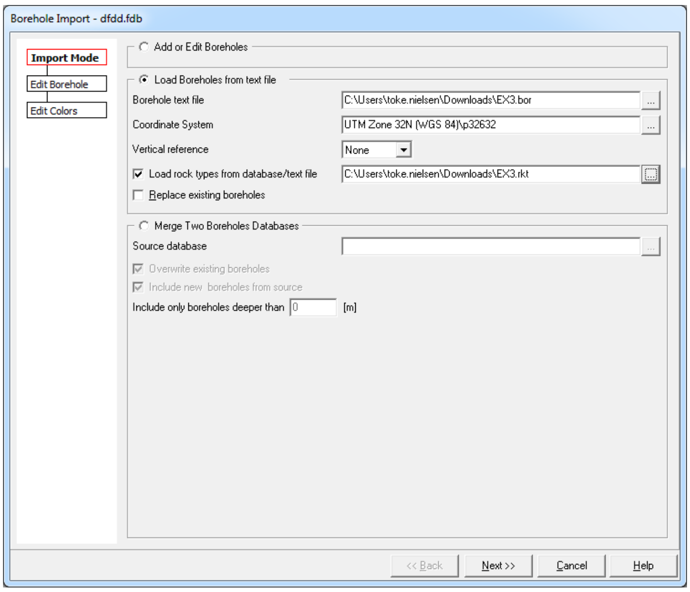
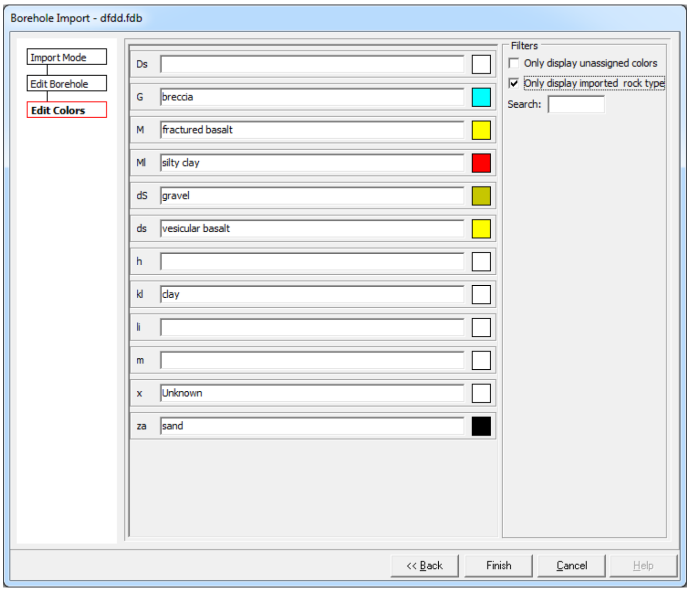


Figure 13

2) Select "Load Boreholes from text file", and select the .bor file. Choose the coordinate system and vertical reference system that fits the borehole data. If no vertical reference system is defined in the .bor file, choose "None" in the dropdown menu. If a .rtk legend file exist, tick the "Load rock types from database/text file" and load the rtk file. Press "Next".



If the .bor file used does not contain the columns RGB and RockDescription, after pressing “Next”, no description and the same white color are associated to each rock type (Figure 3). It is possible to use the editing window in Figure 3 to manually modify the color (by left click on the squares on the right side of the central panel) and the rock description (by typing the description in the editable field next to the rock symbol). Press finish.

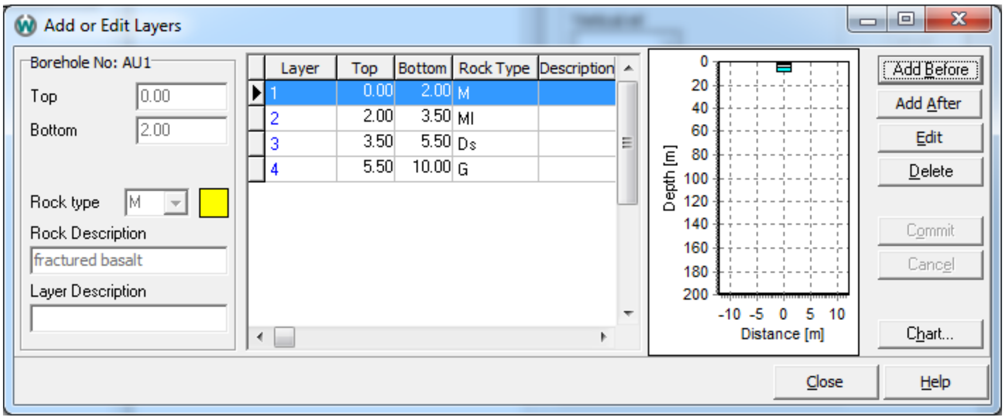


Manual editing of existing borehole information

Existing borehole information can be also edited directly by selecting “Add or Edit Boreholes” option in the import window. Here it is possible to:

- Modify every borehole parameter e.g. UTM coordinates, location, comments, etc.
- Select a symbol representing the borehole in the map (“Map Symbol”).
- Load an appropriate rock type description file (.rkt) by means of “Load rock types”.
- Visualize the existing layer and add new ones (“Add layers”). In the window that pops up after clicking “Add Layers”, the characteristics and the descriptions of each layers can be also modified. Any modification of color and description of a rock type made via this window for a specific layer will be transferred consistently to the same rock type through the entire database.
- Select “Add” instead of “Editing”, then information of a completely new borehole can be entered.
- Once a borehole database has been loaded into Aarhus Workbench, it is possible to modify the rock descriptions and colors directly from the database explorer tree by selecting the borehole database and press “change colors”.

AGS Aarhus GeoSoftware



Import borehole information from existing database

It is also possible to import information from an existing database. This option can be used by ticking “Merge Two Borehole Databases” in the import window and selecting a target database. Then the boreholes in the pointed database will be added (if “Include new boreholes from source” is active) to those already present in the original database. In case of intersection between the information in the two databases, if the option “Overwrite existing boreholes” is selected, in the resulting database the original information will be substituted by the corresponding information in the pointed database.

It is possible to specify the boreholes added to the original database by deciding what their minimum drill depth is. By writing 20 m, only the boreholes deeper than 20 m will be added. The default value is 0 m. This means that, by default, all boreholes are merged into the resulting database.
In the present version of Aarhus Workbench only .fdb databases can be merged.

AGS Aarhus GeoSoftware

Geophysical logs guide

This guide will go through import and display of geophysical logs in the .las 2.0 format. For more information on the format go to http://www.cwls.org/las/

Data examples can be found at: http://www.ags-cloud.dk/Wiki/W_GuidesGeophysicalLogs

.las file criteria's

Both wrapped and unwrapped format is supported. the minimum information needed in the .las file are:

~V /~Version header

Vers.

Wrap.

~W /~Well information

NULL.

X .M (F or FT). Coordinate system has to be UTM.

Y .M (F or FT). Coordinate system has to be UTM.

Z (for elevation - optional)

~C /~Curve information

DEPTH or DEPT (units have to be .M / .F /.FT)

Units

Curve description

~P /~Parameter information

Each line needs to contain a value for each unit

Import

1. Open Aarhus Workbench, open or create a workspace.
2. Go to the "Database Explorer", highlight the "Geophysical data", go to the "Database" ribbon and click "Import".
3. Select Boreholes → Geophysical Log import and choose which database (either existing or new) to import the data into.
4. Add .las files to the importer, select the UTM zone of the UTM coordinates within the .las file.
5. If the UTM zone is different from the workspace, transform the coordinates into the correct UTM zone.
6. Click import

Display geophysical logs

1. Go the “Database Explorer”, highlight the database that contains the data from the geophysical logs. Go to the “Database” ribbon and click New Borehole Selection.
2. A Borehole Log Query will open. Press OK to get all logs selected and give the model selection a name.
3. Go to the Workspace Explorer, expand the MyMap node and tick the box for the model selection. The location of all the logs will be displayed on the GIS map.
4. Go to the “GIS” ribbon and select either one or more logs on the GIS map. Use the pointer or Rectangle tool. The selection can be cleared by the clear button.
5. When the wanted logs are selected click the “Use Selection” button and click “Show data”.
6. A new window for each log selected will be opened.

Keyboard Shortcuts

General

Ctrl + d - New DBQ (database query)
Ctrl + r - Processing settings
Ctrl + f - Properties
Ctrl + p - Show and Edit

GIS interface

Alt + P - Pan
Alt + E - Center
Alt + R - Ruler
Alt + Z - Zoom
Alt + C - Click
Alt + L - Layer
Alt + U - Undo
Alt + O - Redo
Mouse scroll - zoom in and out. Need focus on GIS interface

Chart interface

Alt + P - Pan
Alt + I - Zoom in
Alt + O - Zoom out
Alt + U - Undo zoom
Alt + S - Select point
Alt + key arrows - step in key arrow direction of the chart. Moves x-axis 50% and y-axis 25% of axis-values.
Ctrl + left/right arrow - move chart with buffer size
Ctrl + e - show/hide error bars
Shift + ctrl + v -view data
shift + ctrl + r - view report (only for Jupiter boreholes)

Processing

Ctrl + s - Save edits
Ctrl + u - Update Edits
Ctrl + b - Buffer settings
Ctrl + o - Sounding window
Alt + Q - Disable selected data points
Alt + A - Enable selected data points
Shift + Q - Enable selected IP decay curve(s) - IP only
Shift + A - Disable selected IP decay curve(s) - IP only
Alt + 1 - Give the selected data points 5% standard deviation
Alt + 2 - Give the selected data points 10% standard deviation
Alt + 3 - Give the selected data points 15% standard deviation
Alt + 4 - Give the selected data points 20% standard deviation
Alt + 5 - Give the selected data points 25% standard deviation
Alt + 6 - Give the selected data points 30% standard deviation
Alt + 7 - Give the selected data points 35% standard deviation
Alt + 8 - Give the selected data points 40% standard deviation

Alt + 9 - Give the selected data points 45% standard deviation
Alt + 0 - Reset the standard deviation to the initial value (often 5%)