Workflow for import, processing, inversion and presentation of frequency domain GCM/HEM data in Aarhus Workbench

This guide presents the common workflow options from workspace creation to final presentation of data for GCM/HEM data (data from systems such as DualEM, Geonics EM, Iris Promis, GF Instruments CMD, TerraPlus GEM, and CGG Resolve) 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 on **page 4** 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.

Before the flowchart a short overview of data theory for GCM data in Workbench is included on page 2 and 3.

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.

Instruments

All instruments use electromagnetic induction to estimate the resistivity of the ground. They have one transmitter coil and several receiver coils. There are 3 main things that has influence on the sensitivity of the depth:

- **Distance between receiver and transmitter coils**
  Most systems have a distance of 1, 2 or 4 meters between coils, but others have up to 100m

- **Coil configuration**
  There are many names to describe the coil configurations depending on if you describe the configuration of a coil pair or each individual coil. We describe each individual coil with respect to the moving direction x:

  HMD-x: Horizontal Magnetic Dipole in the x-direction (X).
  HMD-y: Horizontal Magnetic Dipole in the y-direction (Y).
  VMD: Vertical Magnetic Dipole (Z).

  Possible coil pairs supported in Aarhus Workbench:

  HMD-x, HMD-x (x,x) corresponds to vertical coaxial (VCA)
  HMD-y, HMD-y (y,y) corresponds to vertical coplanar (VCP)
  VMD, VMD (z,z) corresponds to horizontal coplanar (HCP)
  VMD, HMD-x (z,x) corresponds to perpendicular (PRP)

- **Operating frequency**
  Some systems are only operating with one frequency and then different spacing and coil configurations. Other has a sweep of frequencies on the same spacing and configuration.

Data

Aarhus Workbench supports 3 types of data units:

- **Apparent conductivity** in **mS/m** with in-phase ratio of the secondary to primary magnetic field in parts per thousand (**ppt**). Inversion is only available for the apparent conductivity, but the phase can be plotted in the processing window.
- **Apparent resistivity** in **ohmm** with in-phase ratio of the secondary to primary magnetic field in parts per millions (**ppm**). Inversion is only available for the apparent resistivity, but the phase can be plotted in the processing window.
- **In-phase** and **quadrature parts**, both in **ppm** or **ppt**. Inversion is available for both parts.

Data in conductivity will be converted to resistivity during import by:

$$\sigma_a = \frac{1}{\rho_a} = \frac{4}{\omega\mu_0 s^2} ppm \cdot 10^{-6}$$

Where s is the coil distance, $\omega = 2\pi f$ where f is frequency and $\mu_0 = 4\pi 10^{-7}$.

Data file format

Aarhus Workbench can import data in xyz format. Non-data lines can be ignored if they have a common sign in the beginning of the line, e.g. "/". This sign can be set in the importer. Lines starting with "LINE" will be used during import and inversion to divide data into lines. Data columns are mapped during import along with all the system settings.

Standard deviations and noise

Both a relative, absolute and user standard deviation can be added to the data, along with a standard deviation from the meaning filters in the processing. The user standard deviation is added if the user manually changes the standard deviation of selected data points in the edit window.

Noise from a GCM/HEM instruments is typically additive compared to the direct measured response which normally is given in ppm. So, to calculate the noise the data values have to be converted into ppm if they are not already) by:

$$ppm_{measured} = \frac{1{,}974 \cdot f \cdot s^2}{\rho_a}$$

And then the total standard deviation for e.g. 5% relative noise and 1 ppm absolute noise can be calculated by:

$$\sigma_{data} = \sqrt{0{,}05^2 + \left(\frac{1\, ppm}{ppm_{målt}}\right)^2}$$

In Workbench we add 1 to the calculated number, so a value of 1.05 equals 5%.

All together the standard deviation for a data point is calculated as:

$$\sigma_{data} = \sqrt{\sigma_{stack}{}^2 + \sigma_{relative}{}^2 + \sigma_{absolute}{}^2 + \sigma_{user}{}^2}$$

NOTE: For data in resistivity or conductivity, a higher absolute standard deviation value gives a smaller calculated standard deviation and vice versa. This is due to the conversion to ppm.

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.

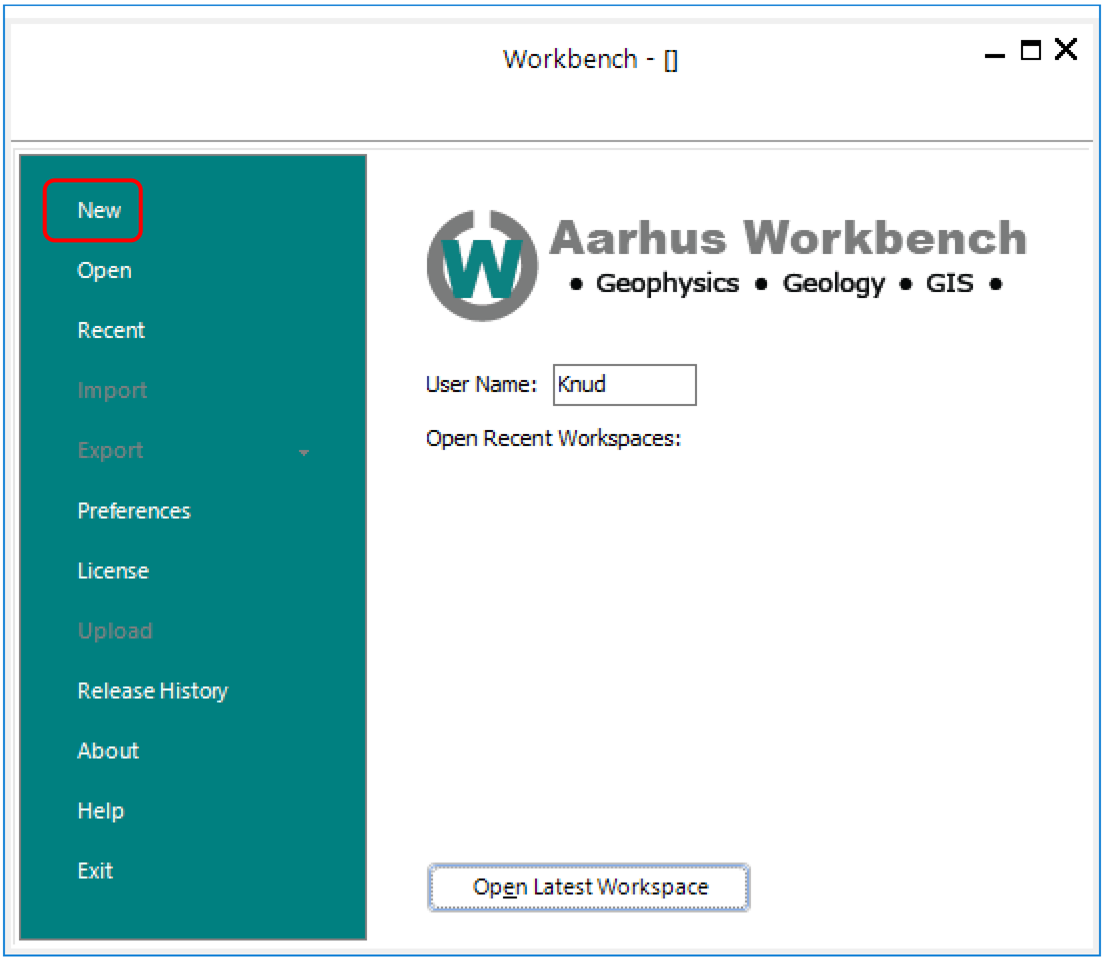


Figure 1 - Creating a new workspace

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.

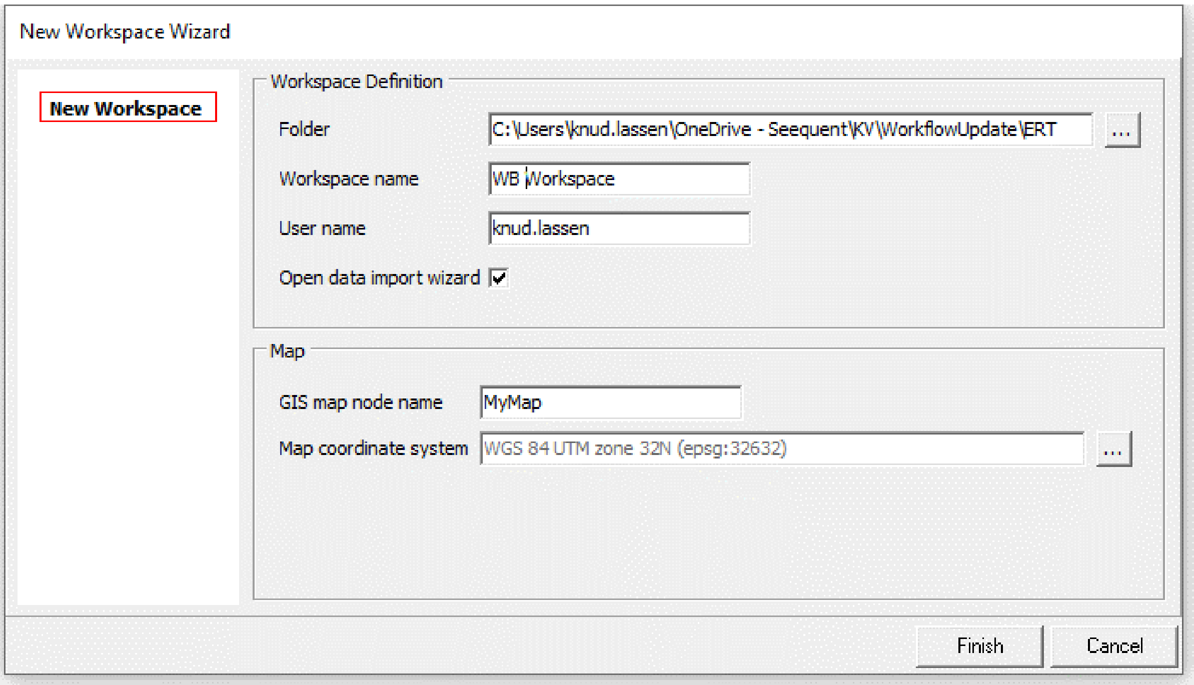


Figure 2 - Setting up the workspace

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.

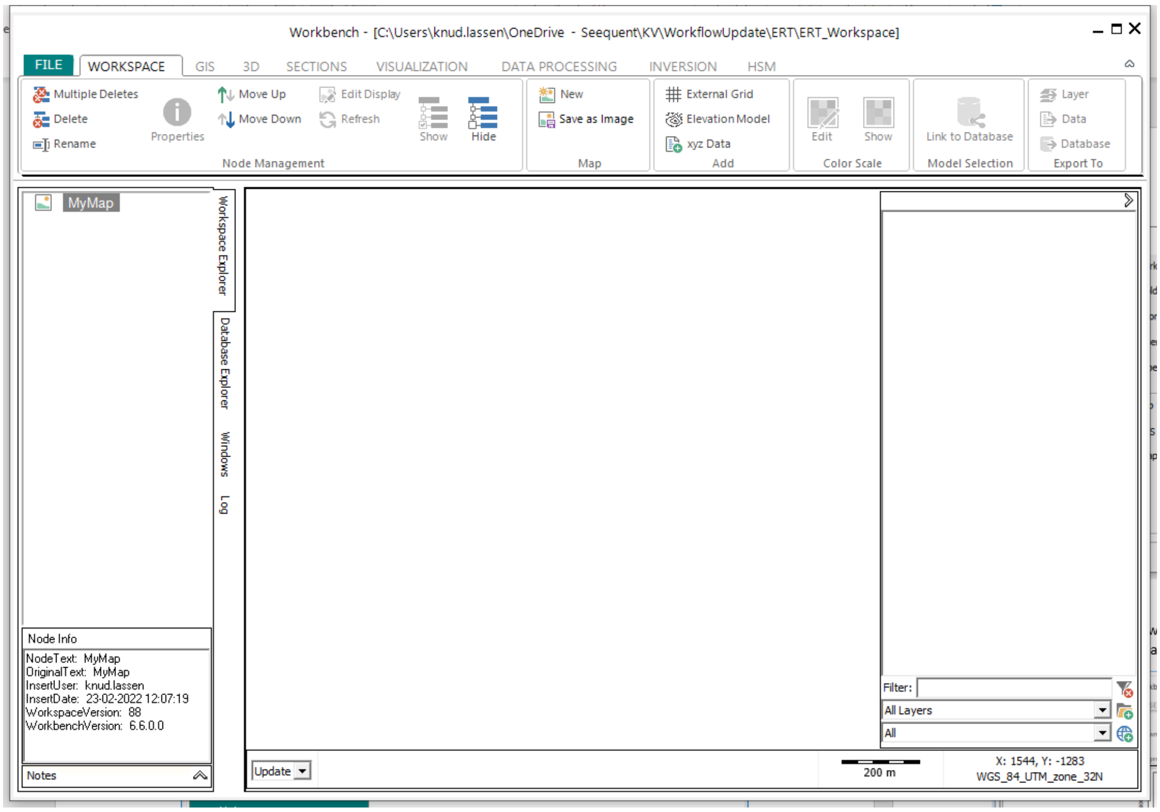
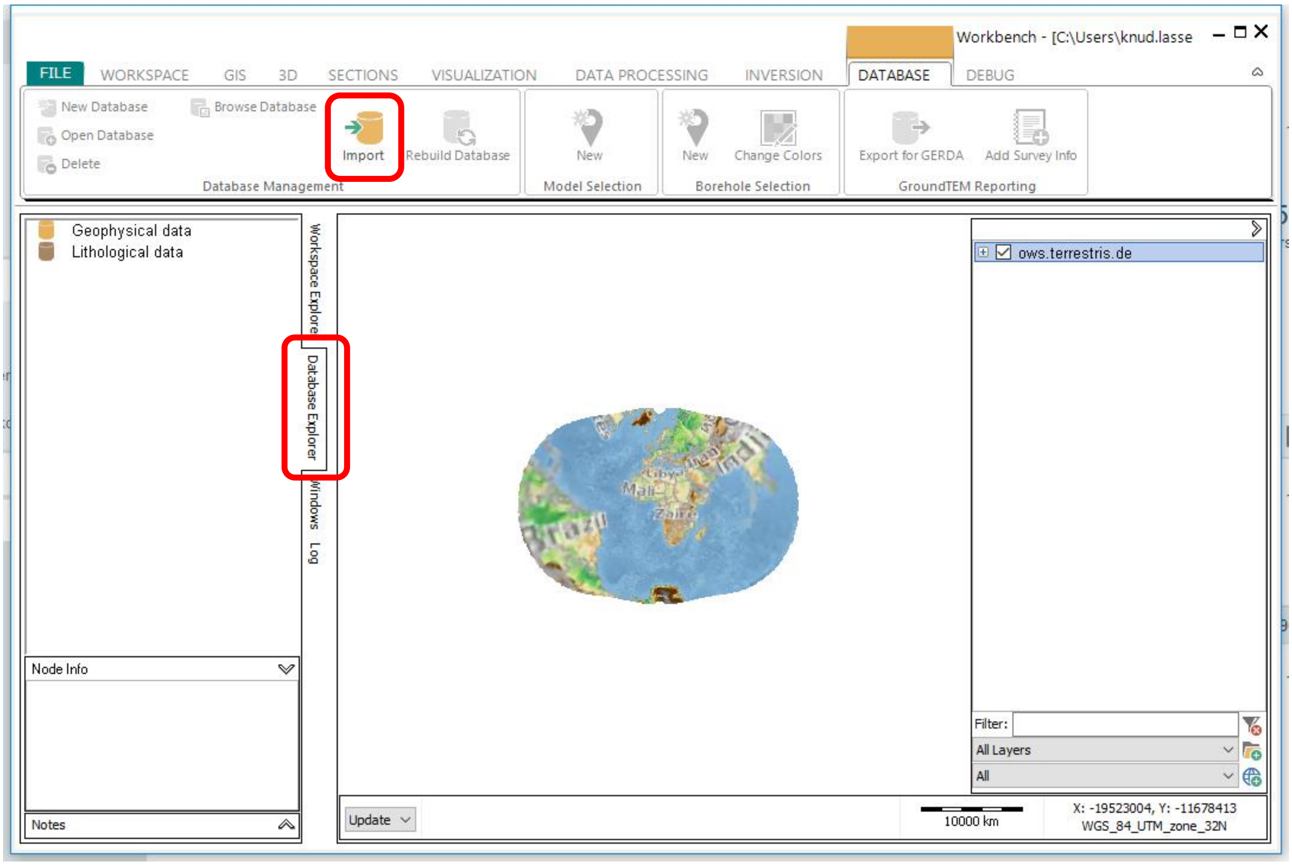


Figure 3 - Initial view of Aarhus Workbench after creating a workspace

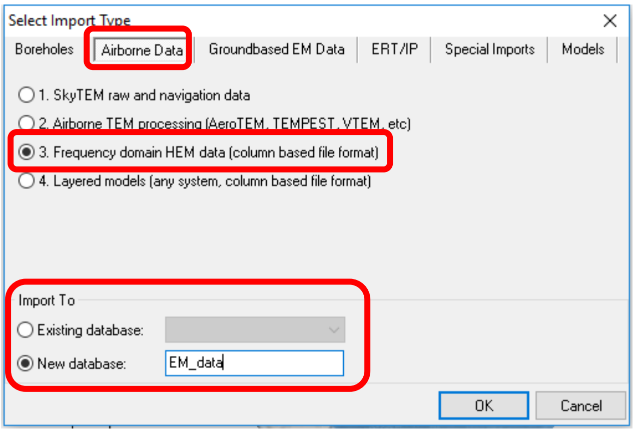
Data import

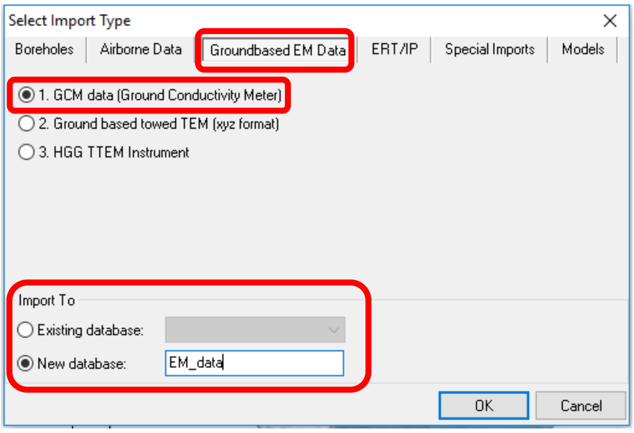
Data import is initialized by selecting the “database explorer” ribbon and pressing “import”:



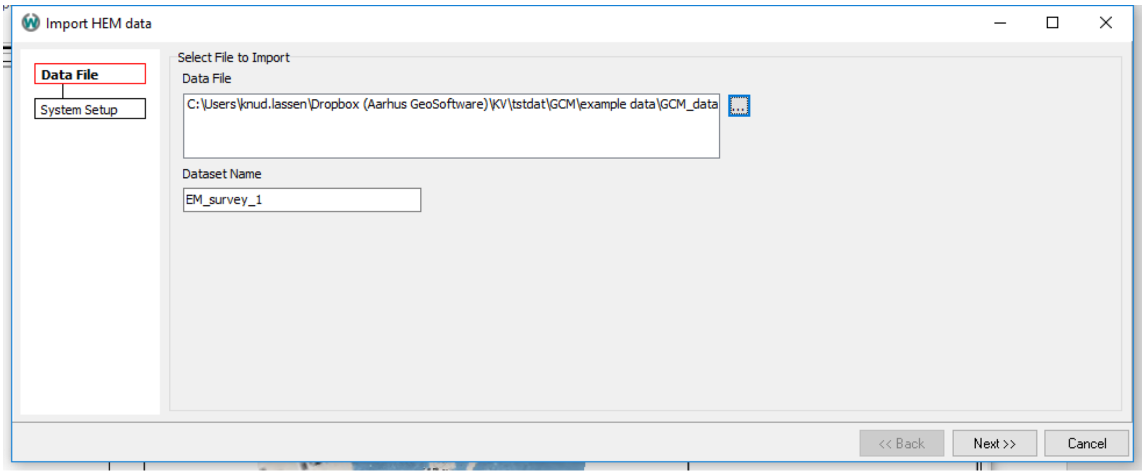
This opens the import window in which either “Airborne Data” or “Groundbased EM Data” data is selected followed by either “Frequency domain HEM data (column-based file form)” or “GCM data (Ground Conductivity Meter)”. If this is the first dataset in the workspace “New database” is selected and a name for the database is entered, if this is not the first import in the workspace an existing database can be selected.

Note: The import procedure is nearly the same for ground based and airborne frequency domain data, the only difference is that the transmitter and receiver height is fixed to only one value for both import and inversion of ground based data, whereas it can vary for airborne data.

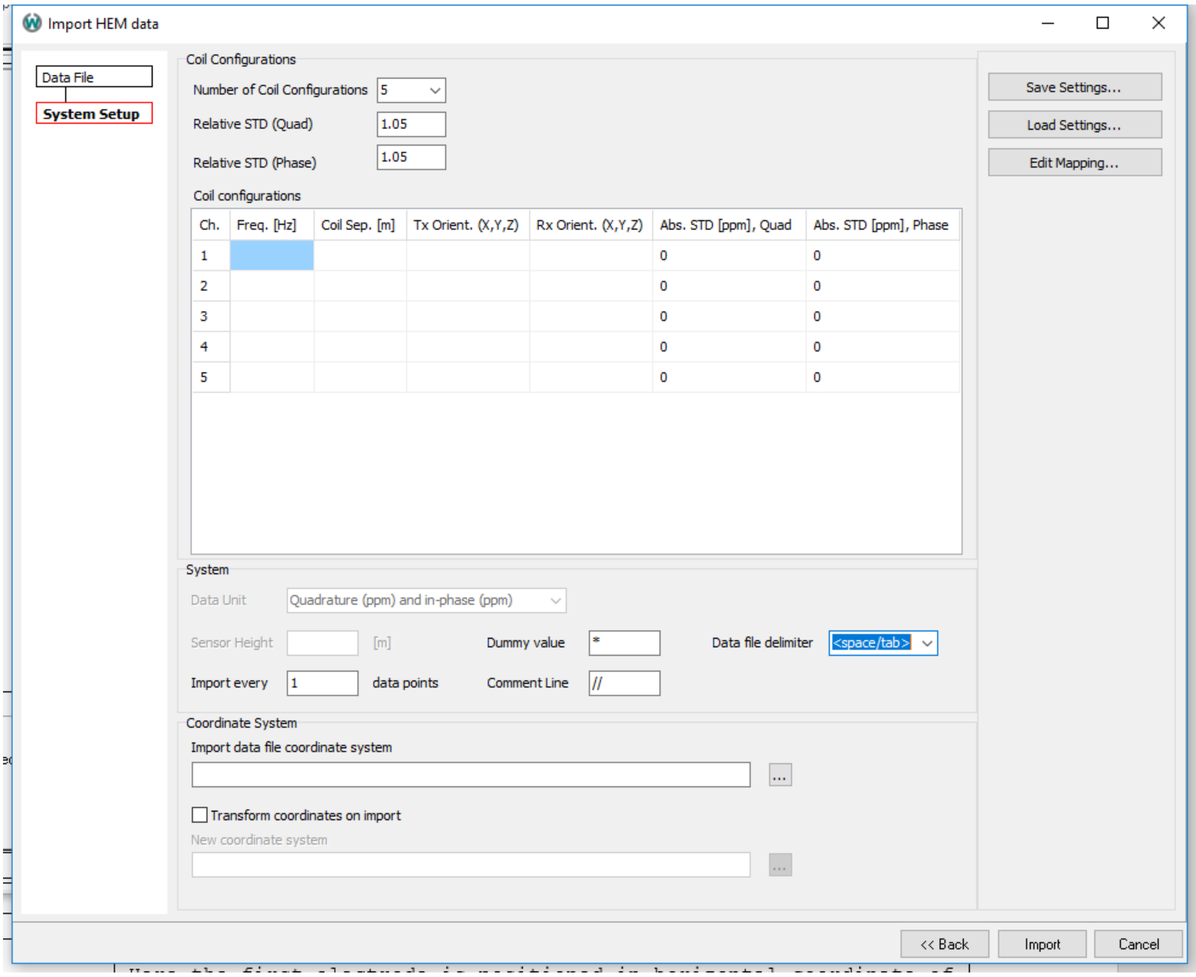




When pressing OK the "Import HEM data" window is shown, in this window the data file for the import is selected, and the dataset is named:

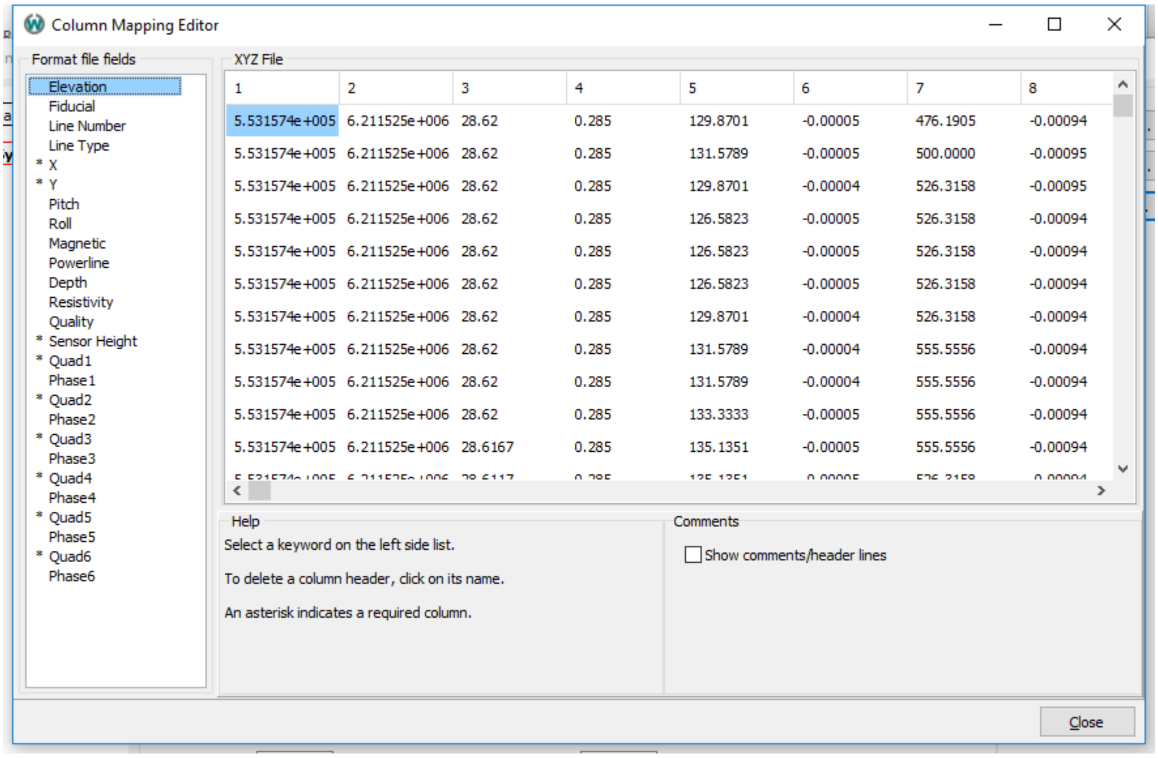


When selecting "Next" the "System Setup" window is shown, here the system configuration and formatting of the data file must be specified:



If this has already been done once, or if a setup file has been provided with the dataset the configuration can simply be loaded by pressing "Load Settings", the format of the configuration file is .htp for airborne frequency domain data and .gtp for ground based frequency domain data. If a configuration file is not provided the properties must be set up manually:

- Specify the system configuration in the "Coil configuration" table, that is, the transmitter frequency, coil separation, Tx (transmitter) orientation and Rx (receiver) orientation, for all data channels. Furthermore, it is possible to specify the standard deviation (STD) of the received signal amplitudes for the quadrature and in-phase signal.
- Specify the data file format in the "system" box by defining the dummy value, delimiter, and comment identifier used in the data file. It is also possible to specify that only eg. every 5th line in the file should be imported. For ground based system the sensor height and data unit must also be specified. For airborne systems only, ppm are supported and the sensor height must be specified in the data file for each measurement. Note that lines starting with "LINE" can be used to divide the dataset into lines.
- Map the individual columns in the data file to the to correct values, e.g. x and y coordinates, data values etc. this is done by pressing "Edit mapping":



-
-

A column is mapped by selecting the property in the box to the left and then clicking the number above the correct column. If the columns in the "column mapping editor" isn't shown correctly it is often due to selecting the wrong data file delimiter. Once the mapping is completed close the "Colum mapping editor".

- Finally select the coordinate system used in the data files and select if the coordinates should be converted to another coordinate system on import (e.g. if the coordinate system is different from that of the workspace).

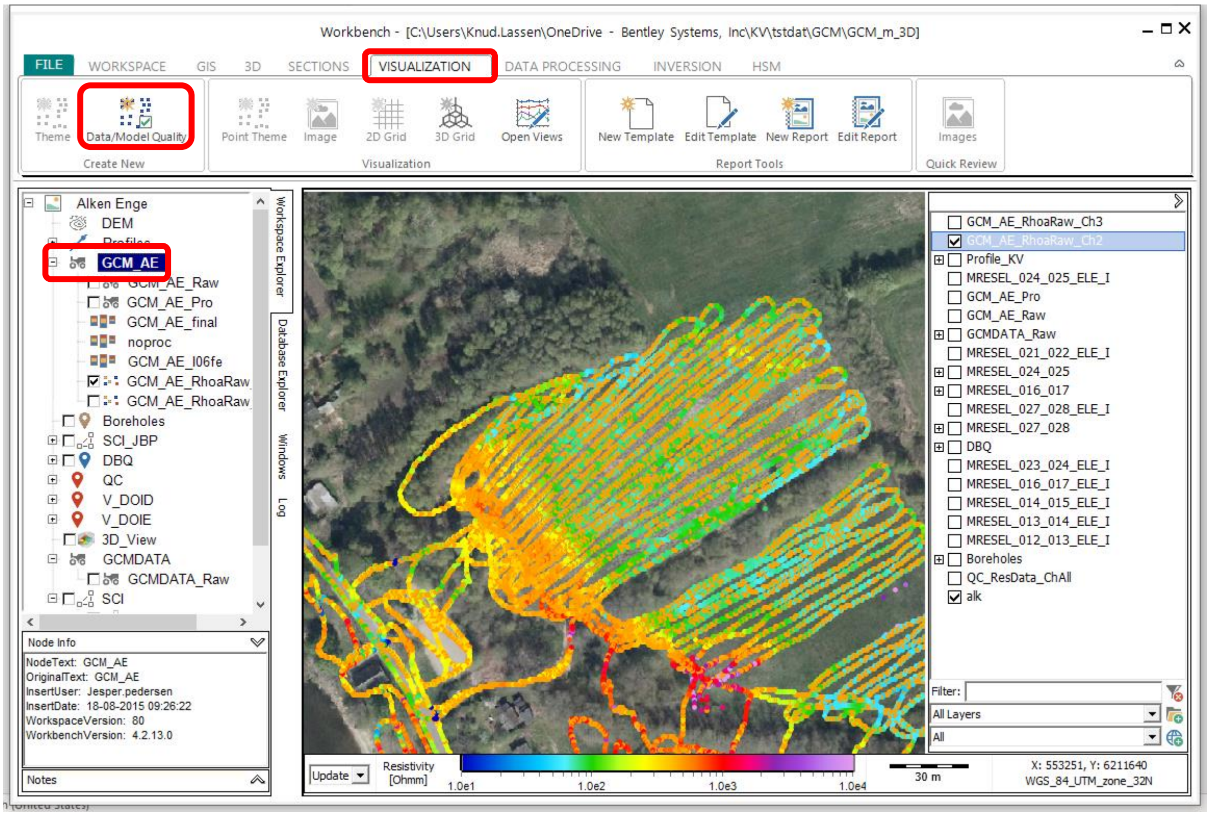
The setup can now be saved by clicking "save settings" to make it faster to import more datasets from the same system. When this is done click "Import" and the data will be imported. Once imported you will be prompted to create a data processing node, select yes, select which map to add the node to (MyMap is default name has been used), select the freshly imported dataset when prompted, press OK, and finally enter a name for the processing node.

New datasets can now be imported, or you can go on to processing and inverting the data.

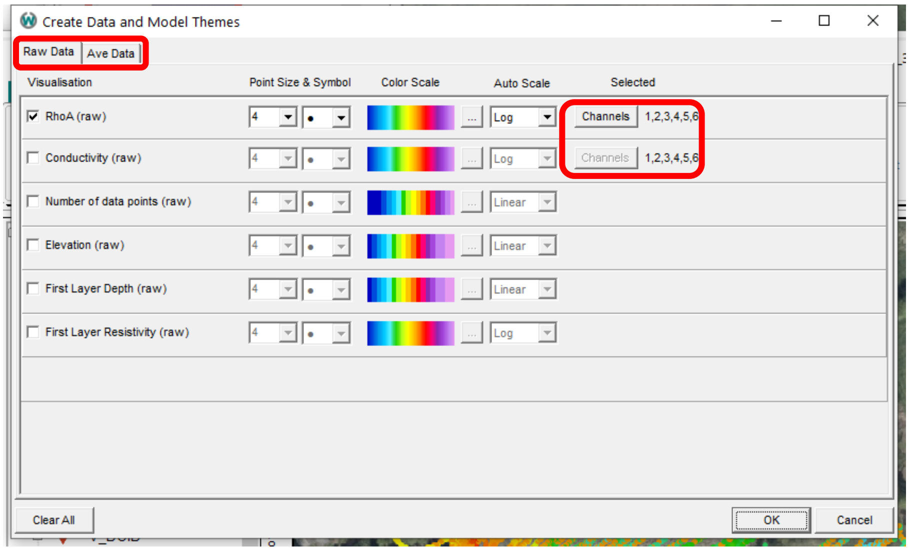
Creating and using data quality control themes

Data Quality themes are used to display different properties of the measured data on the GIS map, either as colored dots or as gridded surfaces. E.g. the value or sign of a measured channel, or auxiliary data such as transmitter current or frequency. This can be done for either raw or averaged data. Such maps are very useful for data evaluation and in some cases for data processing. They can also be used to showcase the quality of the data.

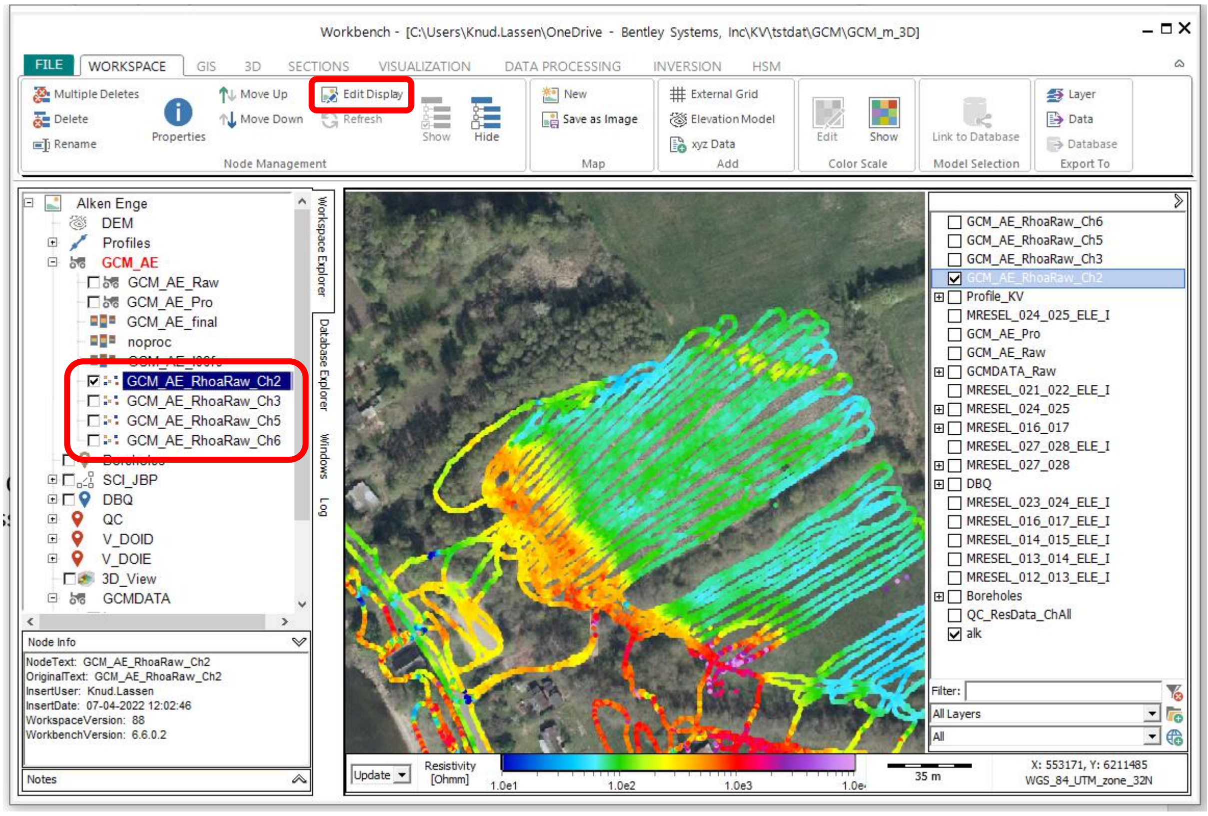
To create a Data Quality node first select a data processing node in the Workspace Explorer and click **Data/Model Quality** on the **Visualization** ribbon:



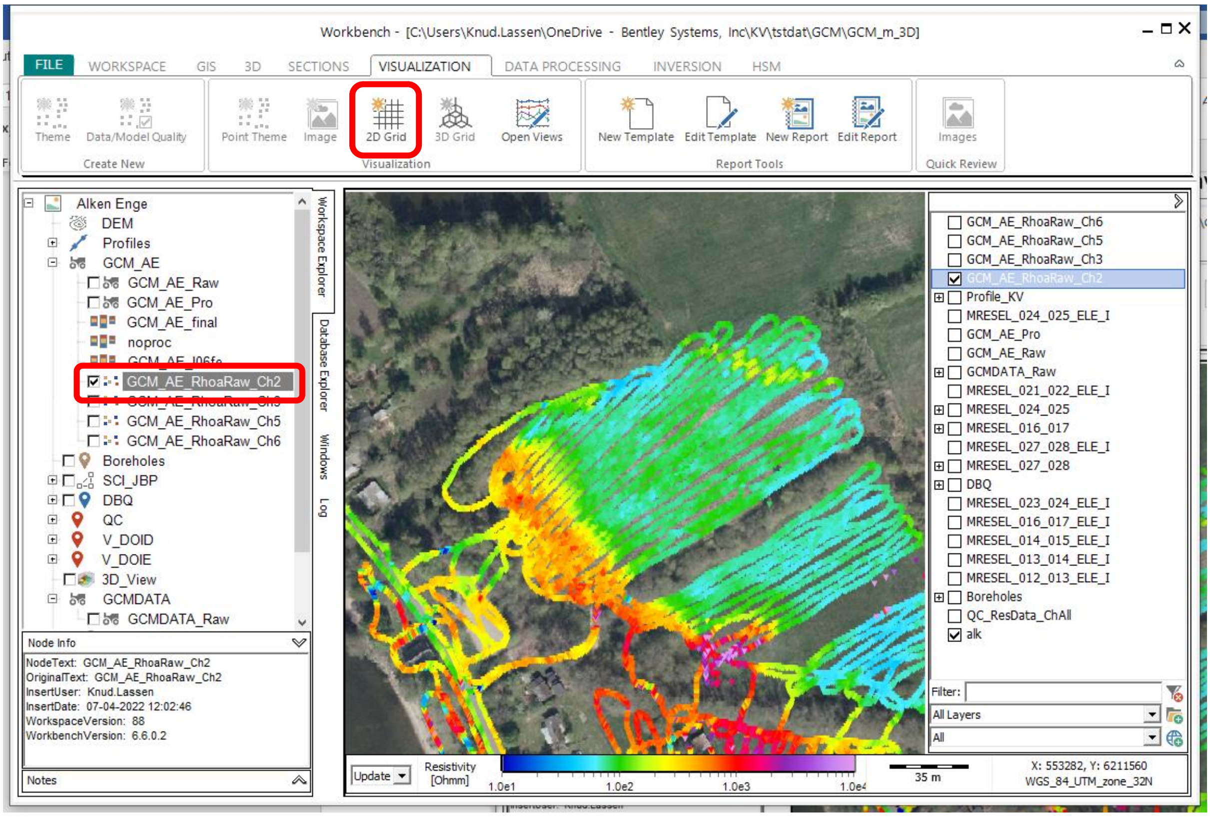
A window with the different options will then be shown, the options will depend on the data type and which parameters have been imported. Check the checkbox next to the parameters that you want to display on the map. Note the option to select specific data channels, and the tab for raw and averaged (processed) data respectively:



Select OK to create the point themes, once these are created they can be found under data processing node in the workspace explorer where they can be toggled on and off. The color scale and point size and shape can be edited by selecting Edit Display on the Workspace ribbon:



To interpolate the point theme into a interpolated map, highlight the point theme and select 2D Grid on the Visualization Ribbon



This will open the grid settings menu where the search radius, resolution and interpolation settings can be specified. Use the F1 help for details on the different settings. Once the grid is calculated select a color scale for the map, the newly generated image can then be found under the point theme node.

AGS Aarhus GeoSoftware

Processing and inversion of GCM/HEM data

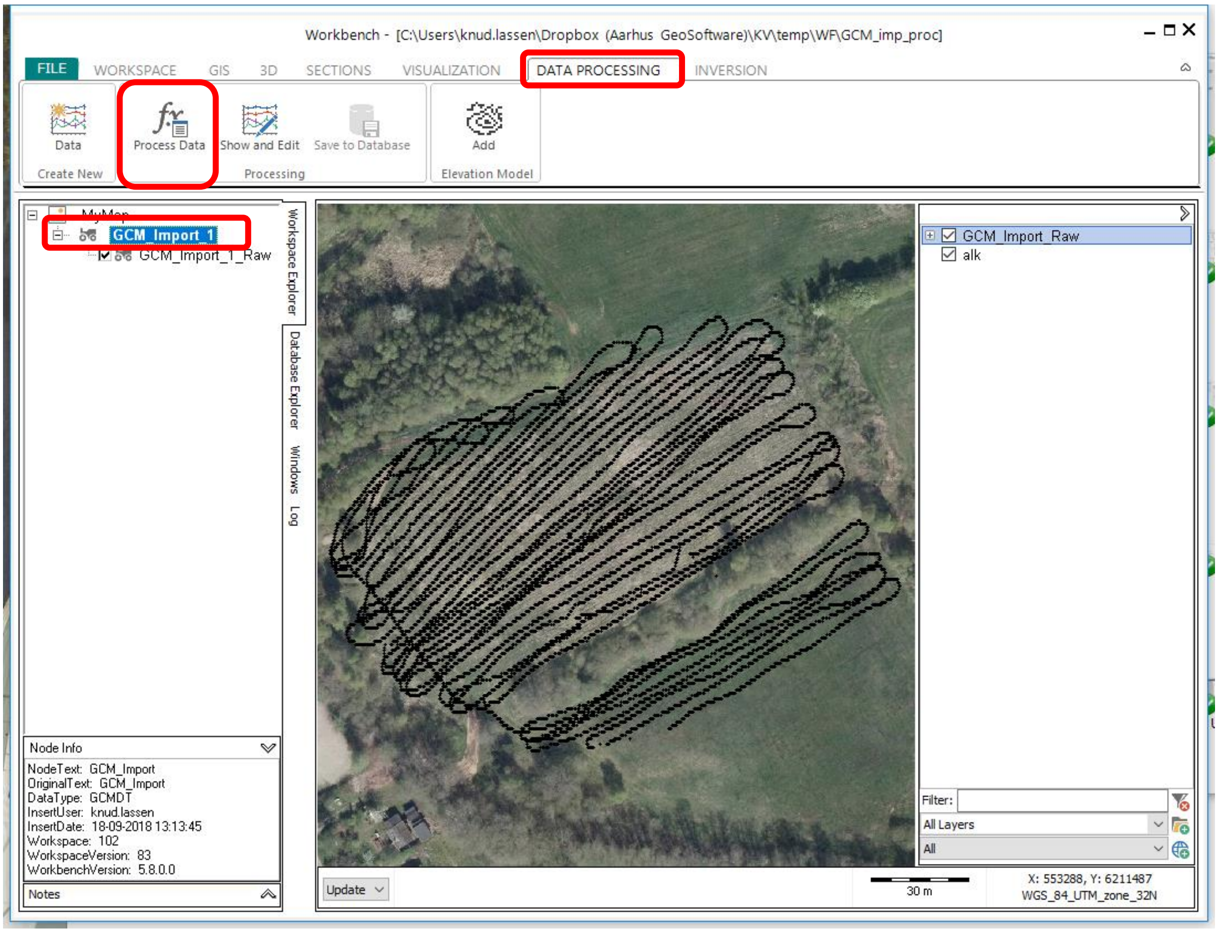
After importing the data and creating the processing node the node can be found in the Workspace Explorer with a subnode holding the raw unprocessed data, by checking the checkbox next to the node the position of the data points will be shown on the GIS map (in this case an aerial photo is used as background map).

Data processing

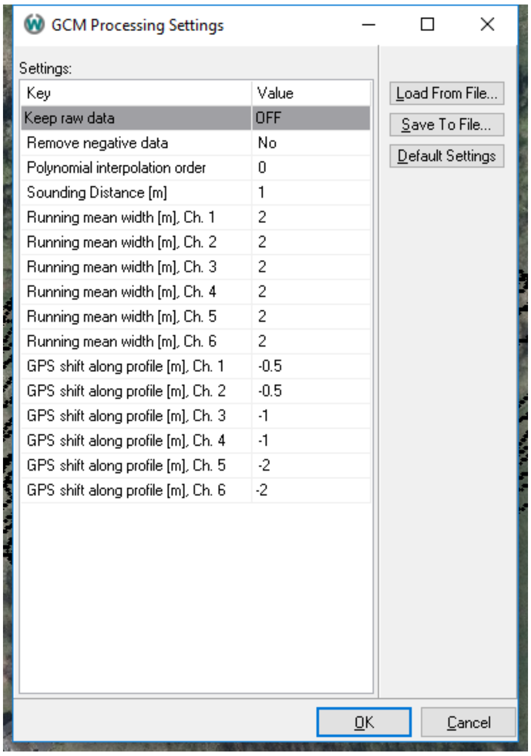
The processing of GCM/HEM data can be divided into two parts, the automatic processing and the manual processing. In the automatic processing the measured raw data is averaged into uniformly spaced soundings, this is done to suppress noise and to optimize inversion time. In the manual processing noisy and coupled data is culled from the survey.

Automatic processing steps

To start the data processing highlight the processing node, navigate to the "Data Processing" ribbon and press "Process Data":



The following window, containing the setting for the automatic processing/averaging, opens:



Once the settings have been created they can be saved to a file for later use by selecting “Save To File” and loaded in again by selecting “Load From File”. The different settings are described in the following:

NOTE, when changing the processing settings, the manual processing will be lost. So it is important to test the different settings, especially the sounding distance and width of the running mean filer, before starting the manual processing as the settings cannot be changed later in the process.

- **Keep raw data**: Keeps the raw data from import. All the below processing options are neglected.
- **Remove negative data**: Negative data will not be removed automatically if the set to “No”. To use negative data, the inversion has to be run in linear space which can be changed during the inversion setup.
- **Polynomial interpolation order**: With a setting of 0, the processing uses the running mean instead. With a value of 1 or more it tries to fit the data to a polynomial of order 1 or 2 etc. In general, we suggest using the running mean.
- **Sounding distance [m]:** Sets the sounding distance for the processing i.e. the distance between the soundings after averaging. The length should be more than the average sounding distance of the measured raw data.
- **Running mean width [m]**: sets the width of the running mean filter for each channel. The value has to be small enough that geological features are not smeared out, but big enough to suppress noise in data. This setting is often changed several times before the right balance for the given dataset is found. Each channel can have a different value since some channels can be noisier than others, and due to the more averaging nature of coil configurations with deeper focus points.
- **GPS shift along profile [m]**: Relative position of the different channels along the profile relative to the GPS receiver, the position of the channel is usually taken as the midpoint

between the transmitter and the receiver. The positive direction is the walking/driving/flying direction.

Select “OK” to apply the settings to the data. A new subnode is added to the data node with “_pro” added to the name.

To assess the effect of the automatic filtering and other processing settings highlight the data node in the workspace explorer and select ‘Show and edit’ in the ‘Data processing’ ribbon. The view window shown in Figure 4 will then open. Here you can see ram unprocessed data for all channels in the uppermost “Rhoa raw” plot, and the same channels after processing below in the “Rhoa ave”. The goal is to strike the balance between averaging as much noise as possible out of the data without smearing out the details of the geological features. The view can be modified in an number of ways, use the F1 help for further details.

Manuel processing steps

Once satisfying settings for the automatic processing have been applied it is time to move on to the manual processing. Again, please note that the processing settings SHOULD NOT be changed after the manual processing has been started, as this will overwrite all manual processing.

To begin the manual processing highlight the data node in the workspace explorer and select ‘Show and edit’ in the ‘Data processing’ ribbon. The edit view will now open:

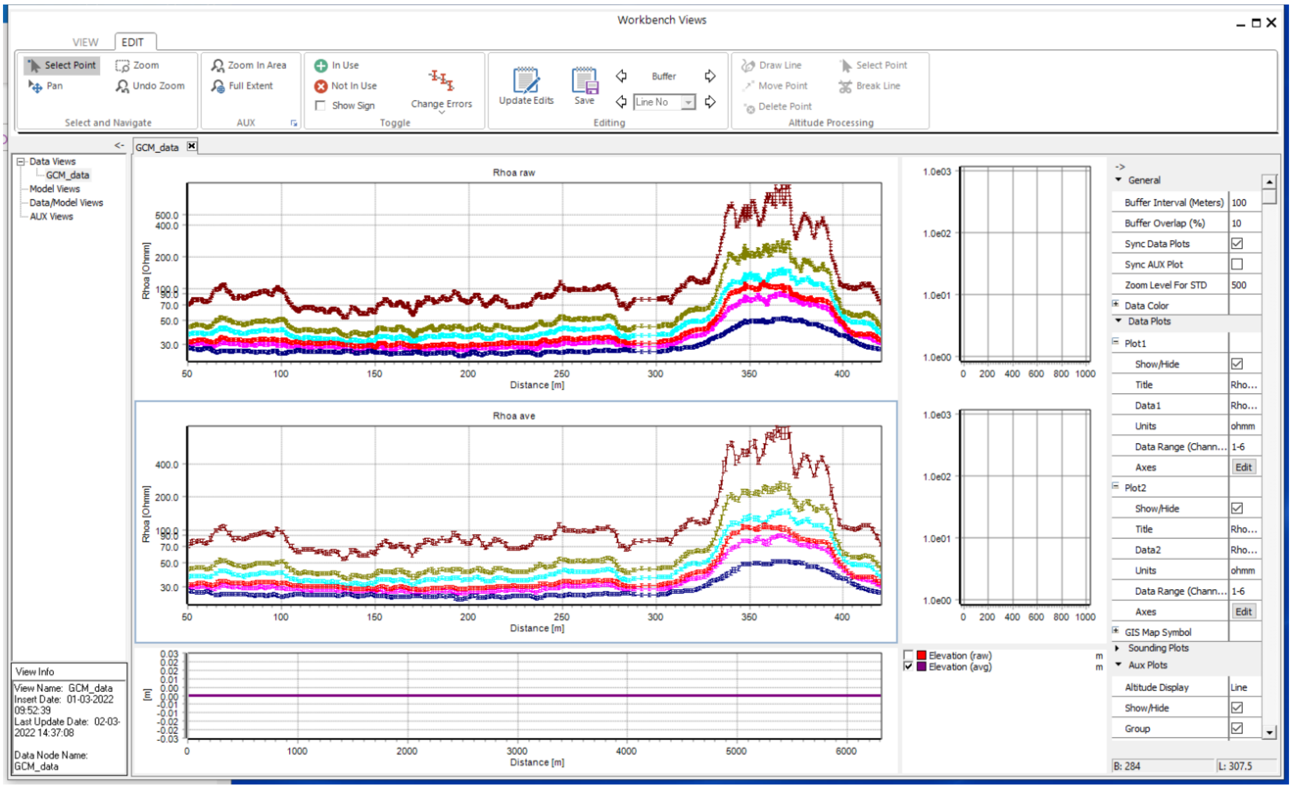


Figure 4

This view contains a number of plots to aid the processing in the best way:

- The raw and average data plots as already described

- The "Aux" or auxiliary plot below the "Rhoa ave" plot, which can be used to show additional information recorded during the survey. In the example in Figure 4 is just shows the elevation of the instrument above the surface, which is constant as this is a ground based GCM survey. But examples of information that can be displayed here is transmitter and receiver temperature, distance above ground and everything else that is recoded during the survey. This can be used during processing to e.g. remove data where some irregularity is detected in the equipment.
- The sounding plots to the right of the raw and average plots. Soundings can be added to these plots by drawing a square in one of the data windows while holding down the shift key.
- The amount of data to be shown at the same time can be changed by changing the "Buffer Interval" in the menu to the right.
- Navigate through the entire dataset by pressing the arrows on either side of the "Buffer" button on the "Edit" ribbon.

The aim of the processing is to go through the entire dataset and remove all datapoints affected by too much noise as well as couplings to non-geological features such as manmade structures. If the instrument is towed behind a vehicle it is also common that the vehicle results in couplings while turning because the distance between the instrument and the vehicle decreases.

Figure 4 above shows a classic example of coupled data between approx. 300 and 450 meters. To remove data select the points by holding down the left mouse button and drag the box around the data, once the points are selected they can be removed by pressing **Alt+Q** to reactivate points press **Alt+A**. Most of the manual processing is usually done in the raw data, because it is easier to see outliers and couplings before they are smeared out by the averaging. And because you can keep more data points this way, as the averaging mixes the couplings and outliers into the signal from the good data points, so more points would have to be removed if processing were done in the averaged data. To see the effect of the processing from the raw data in the averaged data press "Update edits".

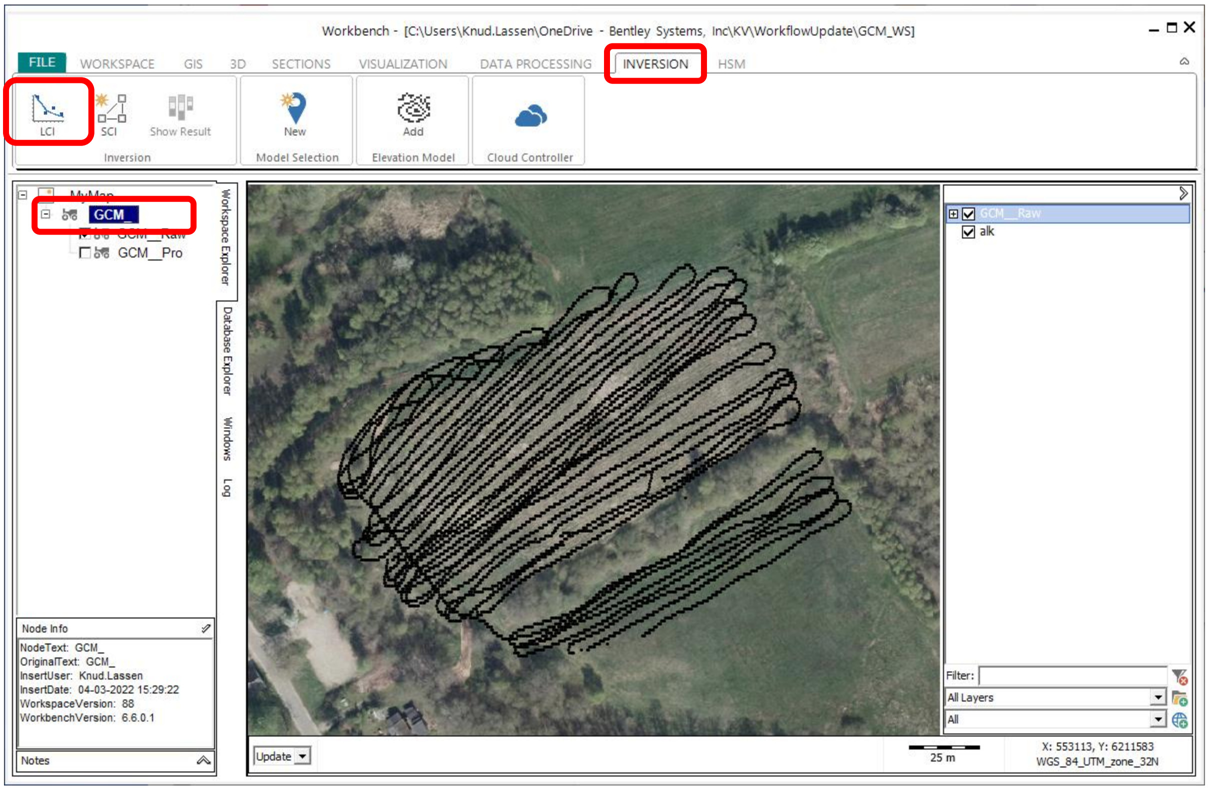
In addition to entirely removing data points it is also possible to increase the standard deviation on noisy parts of the data by selecting the point at pressing **Alt+2** (for 10% standard deviation), **Alt+3** (for 15% standard deviation) or **Alt+4** (for 20% standard deviation) etc. For a full list of processing keyboard shortcuts see the end of this document.

Note that while the pointer is moved around in in processing window a red dot is shown on the GIS map to show the exact location of the data. This can be of great help when trying to determine if data are affected e.g. by a road or a fence, and to find the parts of data that are affected by turns. It is also possible to go from a location on the map to the exact point in the data window, this is done by selecting "Find nearest" in the GIS ribbon and clicking the location in the map.

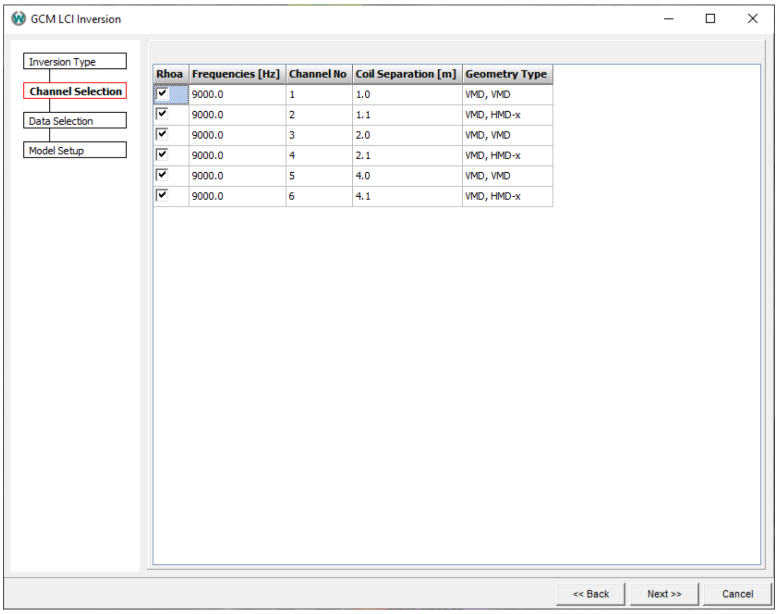
Once the processing is done the data can be inverted and possibly reprocessed. Reprocessing is usually a question of finding couplings and noisy data that was missed in the first processing round but becomes apparent due to poor data fit (see: Quality control themes) or artefacts in the inversion.

Running a GCM/HEM laterally constrained inversion (LCI)

A GCM/HEM inversion in run by highlighting the inversion node and pressing "LCI" in the "INVERSION" ribbon:

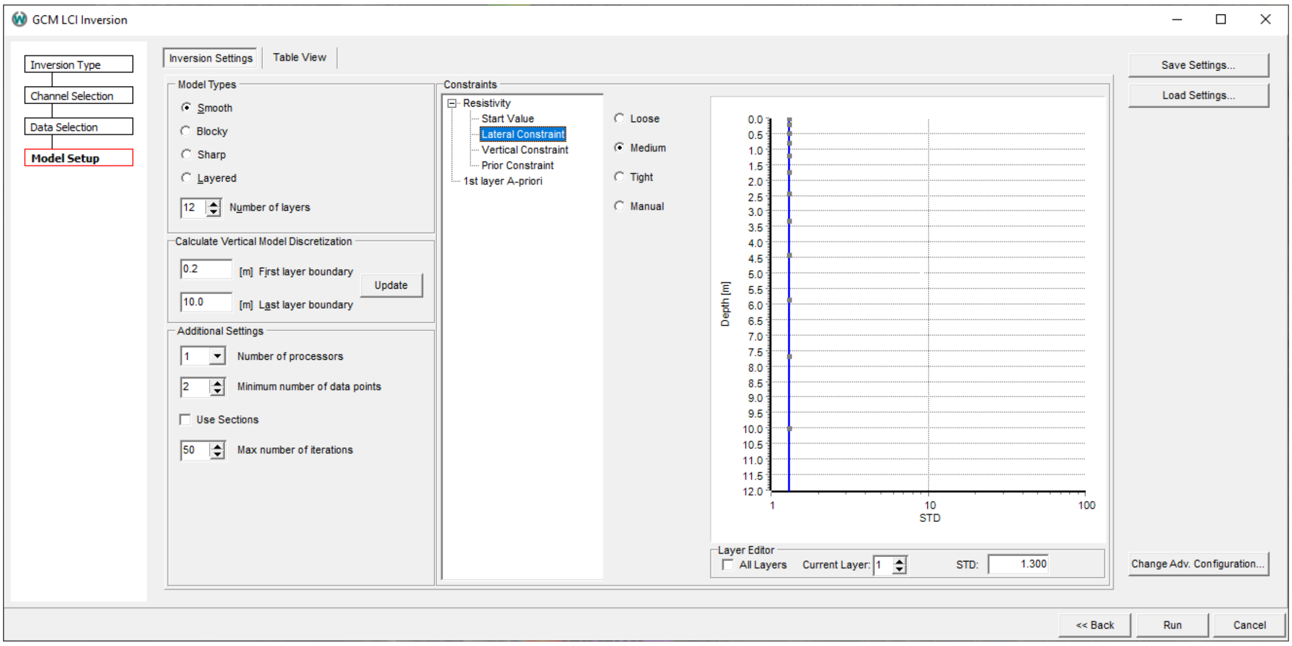


The inversion wizard is opened in which the name of the inversion must be entered on the first page, when selecting "Next" the following menu is shown in which the channels/configurations to use in the inversion are selected:

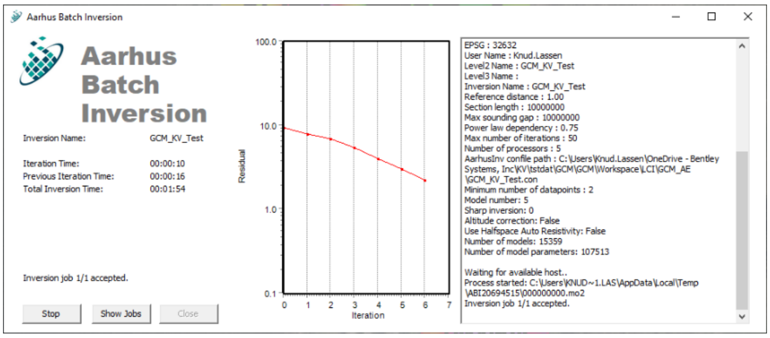


Rhoa	Frequencies [Hz]	Channel No	Coil Separation [m]	Geometry Type
☑	9000.0	1	1.0	VMD, VMD
☑	9000.0	2	1.1	VMD, HMD-x
☑	9000.0	3	2.0	VMD, VMD
☑	9000.0	4	2.1	VMD, HMD-x
☑	9000.0	5	4.0	VMD, VMD
☑	9000.0	6	4.1	VMD, HMD-x

In the next window it is possible to select whether to invert the entire dataset or only some parts of it. Finally the inversion setup window is reached in which a number of settings for the inversion are defined, use the F1 help for a full explanation of all the settings:



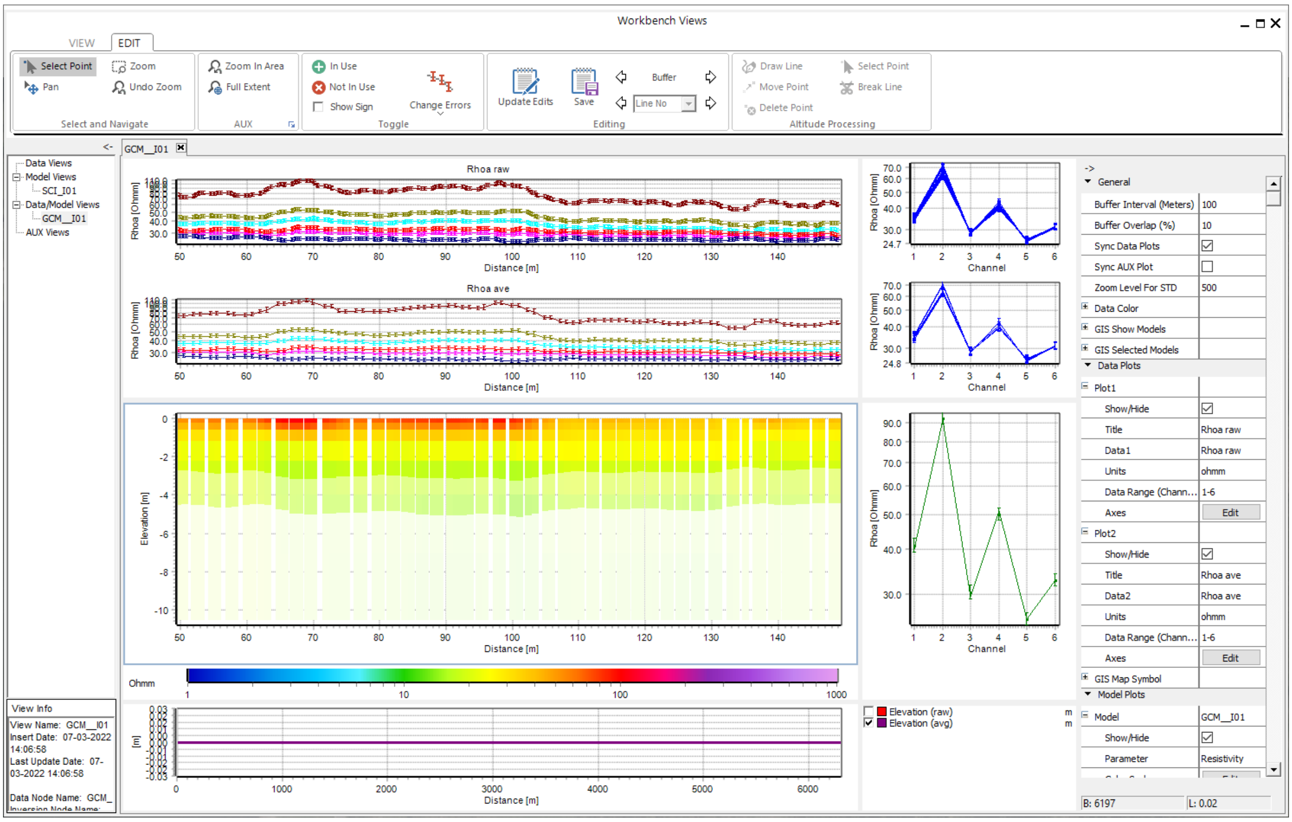
Press “Run” to run the inversion with the selected settings. The progress can be tracked in a separate window:



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.

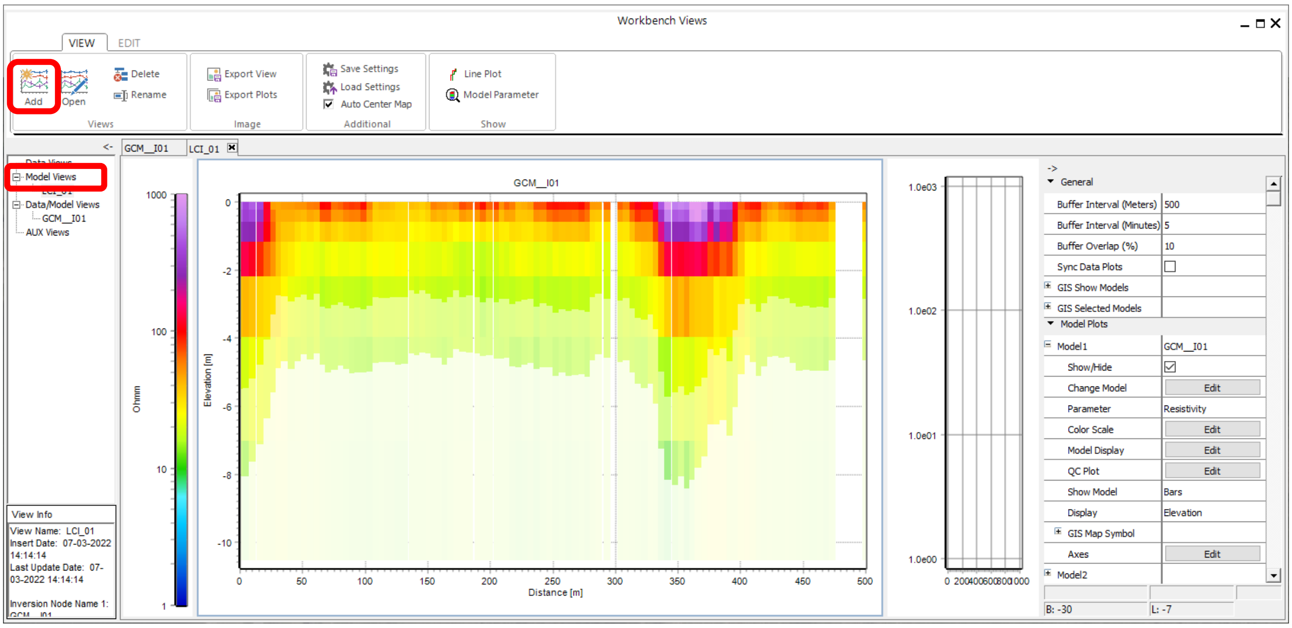
Evaluating inversion results

Once an inversion has been carried out the results can be examined by highlighting the inversion node in the workspace explorer and selecting “Show Result” in the “Inversion” ribbon, the following view will then be displayed:



The layout and navigation is exactly the same as for the manual data processing view encountered earlier, but with the addition of the inversion results to the AUX and data plots. This is a very useful view to make sure that all couplings have been removed from data, and if not, the data can be reprocessed from this view before reinversion. Note that all seletions are also shown on the GIS map at all times.

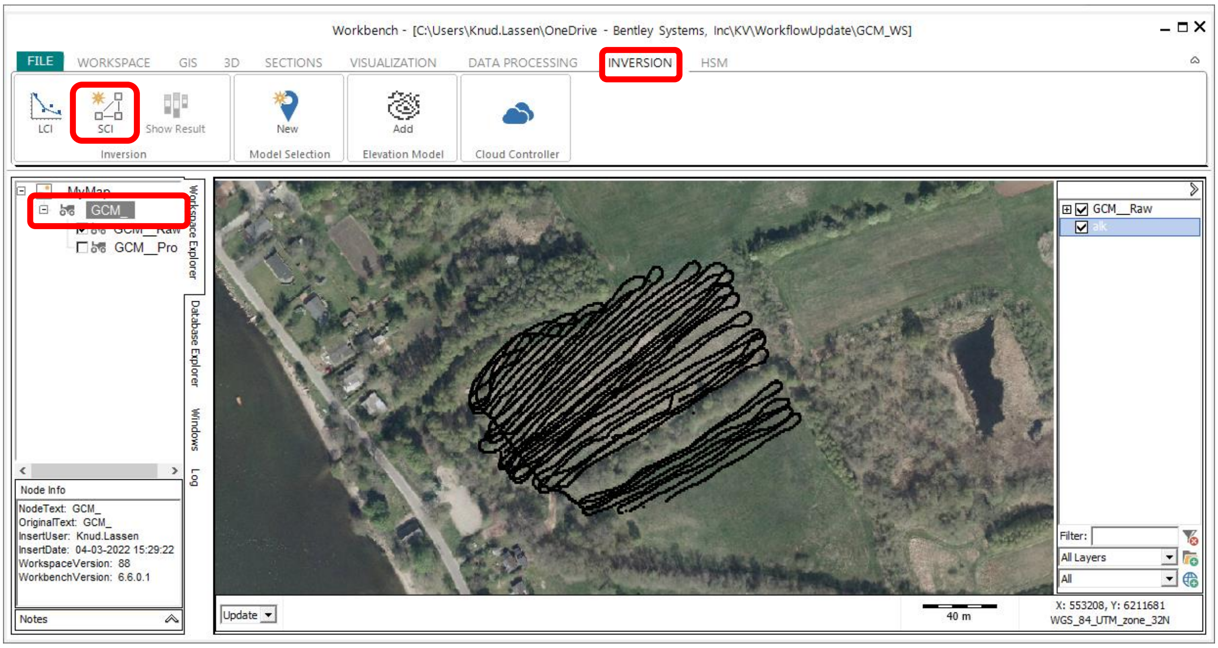
It is also possible to create a view only showing the inversion results without the other plots, this is done by highlighting "Model Views" in the left menu, and the selectio "Add" on the "VIEW" tab:



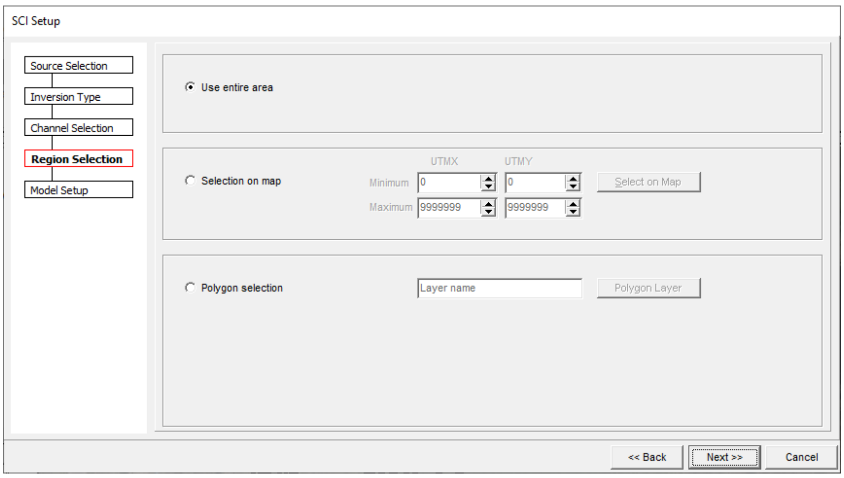
Running a GCM/HEM spatially constrained inversion (SCI)

The models in a LCI is only constrained to ensure continuity along the driving direction, a SCI is constrained in all directions, this makes sure that there is no ”preferred direction” for structures in the inversion, and gives a more coherent pseudo 3D inversion result. The tradeoff is that the inversion takes longer time to run than the LCI. Often LCI’s are used as preliminary inversions to aid data processing and test different settings, and then a final SCI is run that is used for visualization and interpretation.

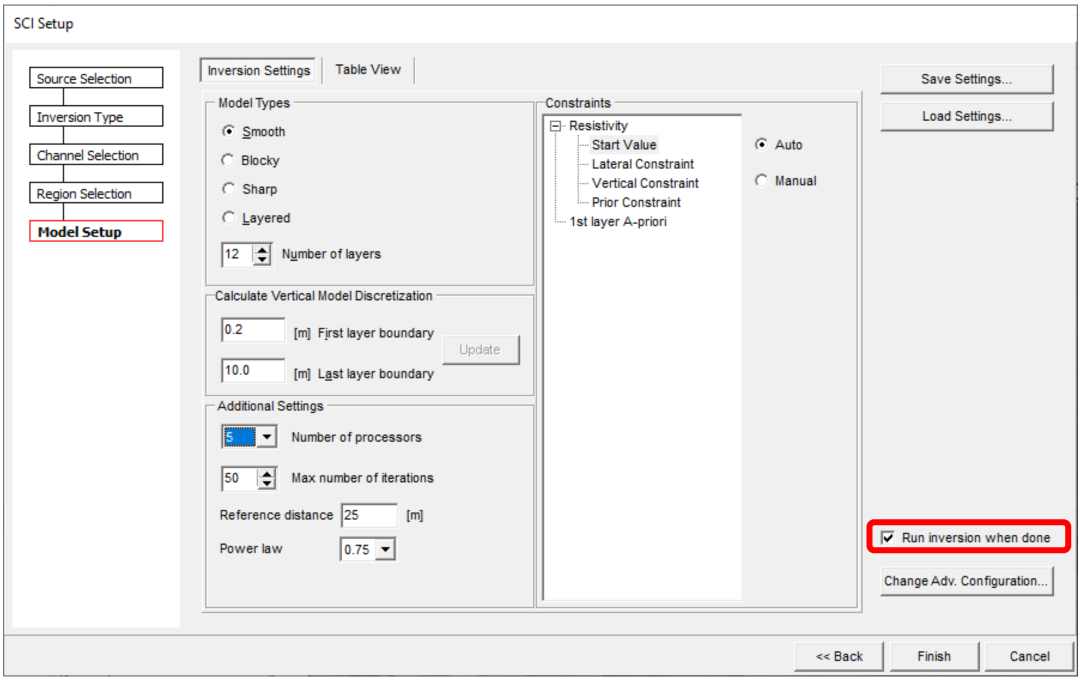
To run a SCI inversion, highlight the data node in the workspace explorer and select “SCI” from the inversion ribbon:



In the following window the data source must be selected, that is the GCM/HEM data nodes that we want to include in the inversion. Once the data is selected press “next” and specify a name for the inversion, press “next”, select the channels for the inversion in the same way as for the LCI, select “next”. In this window it is possible to chose between inverting the entire data node, or to select the area to be inverted either from the map or using a GIS polygon:

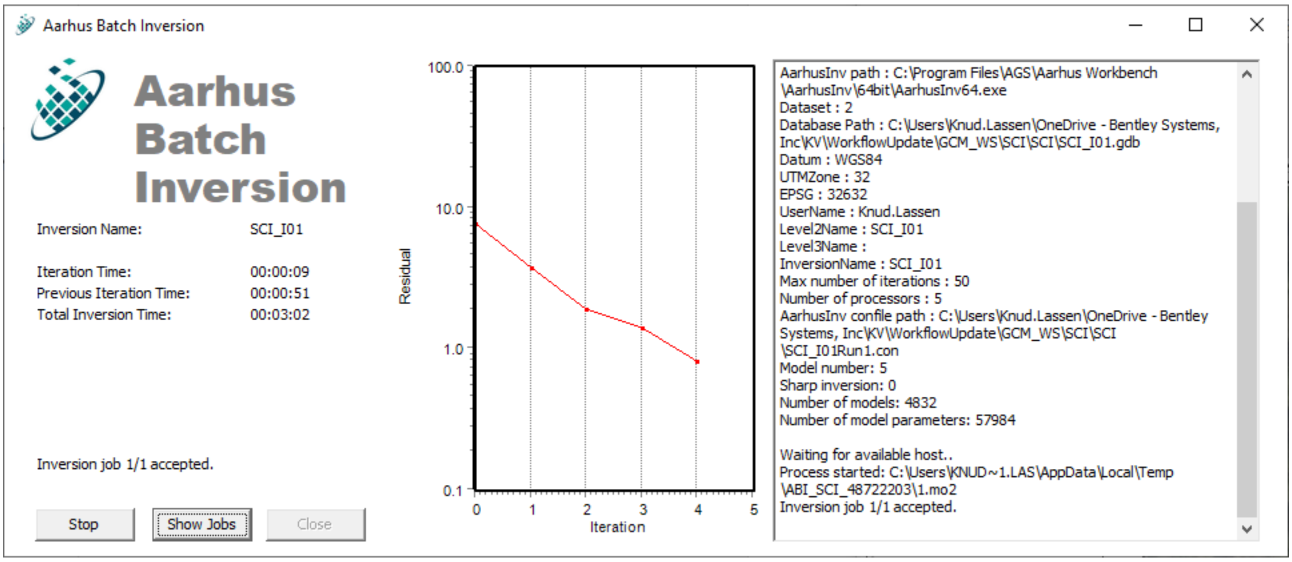


In the next window the inversion type and constraints is selected in the same way as for the LCI, for a detailed description of the individual options press F1:



When selecting "Finish" the inversion will launch if the "Run inversion when done" checkbox is checked, otherwise it can be started by highlighting the new inversion node in the workspace explorer, and then selecting "Invert Data" on the "Spatially Constrained" ribbon.

The status can be followed from 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.

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 5) 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.

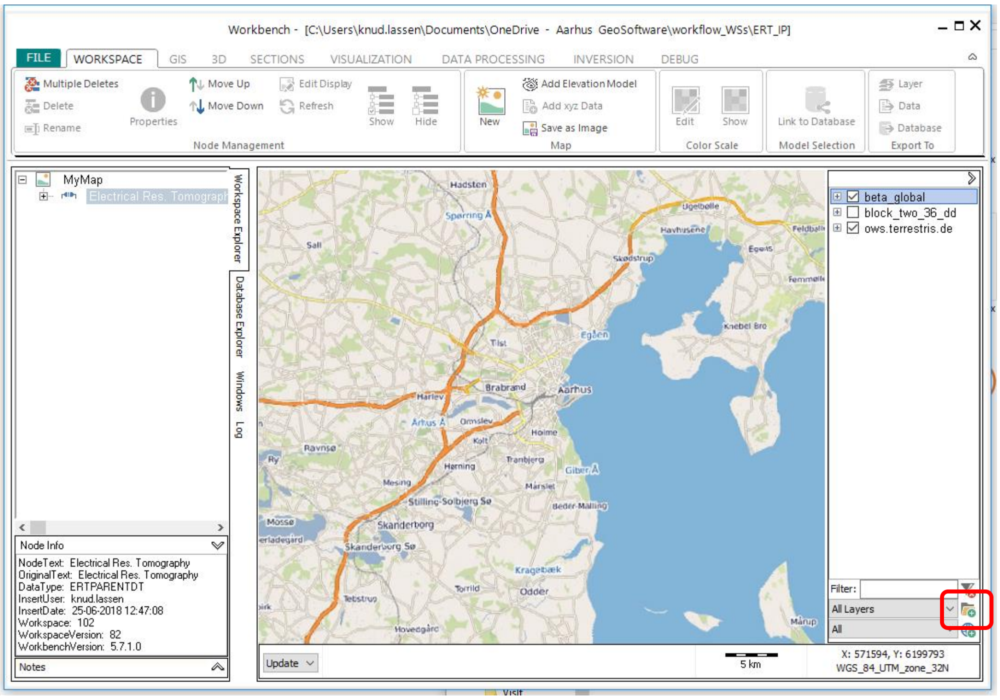


Figure 5

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 5, 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 colour scale on the GIS maps.

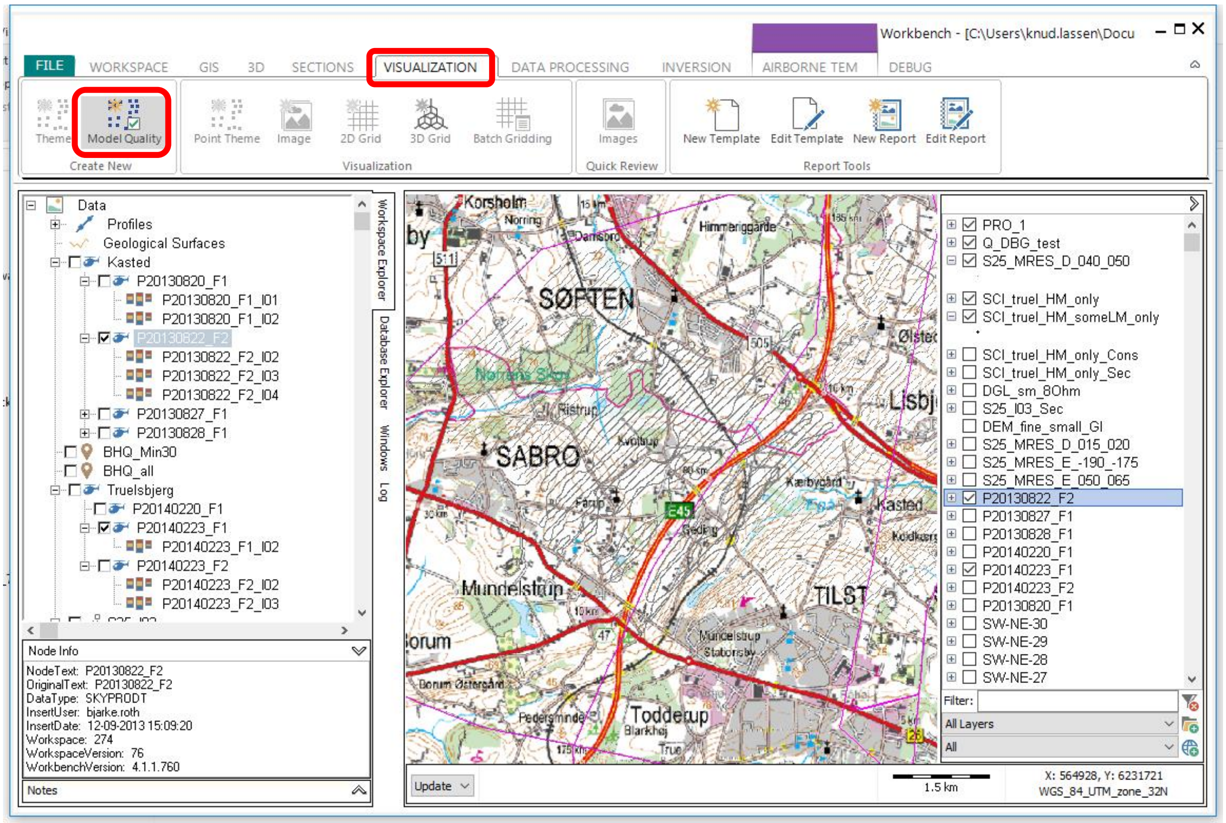
Common values to plot for the QC is total residual, data residual and dept of investigation (DOI).

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.

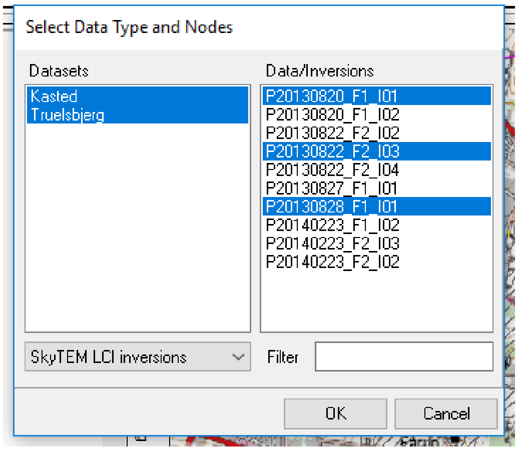
DOI can be used to determine whether the survey resolve the subsurface to a sufficient depth to serve the surveys purpose, and to guide the interpretation of data.

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.

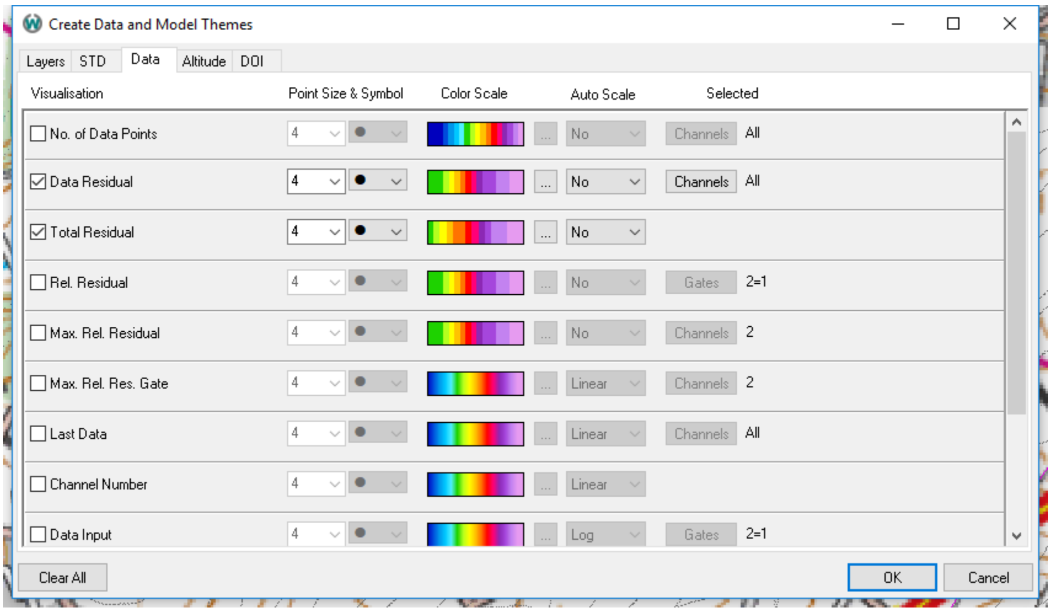
Model quality themes are created by navigating to the "Visualization" ribbon and selecting the "Model Quality" button:



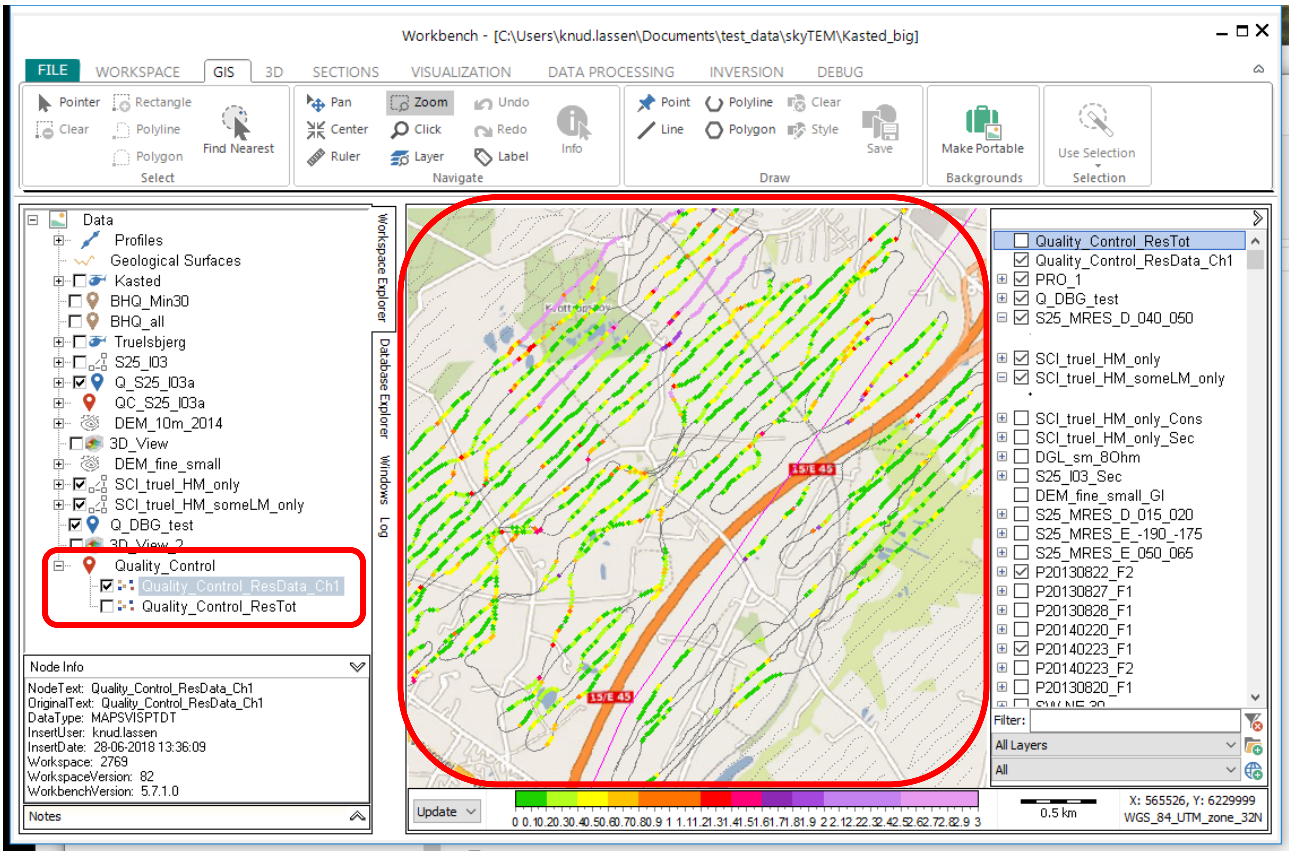
In the resulting dialogue box, it is possible to select which inversions from which datasets to work with. Note that more than one dataset and inversion can be selected by holding down "CTRL" while selecting:



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 coloured symbol on the GIS map:

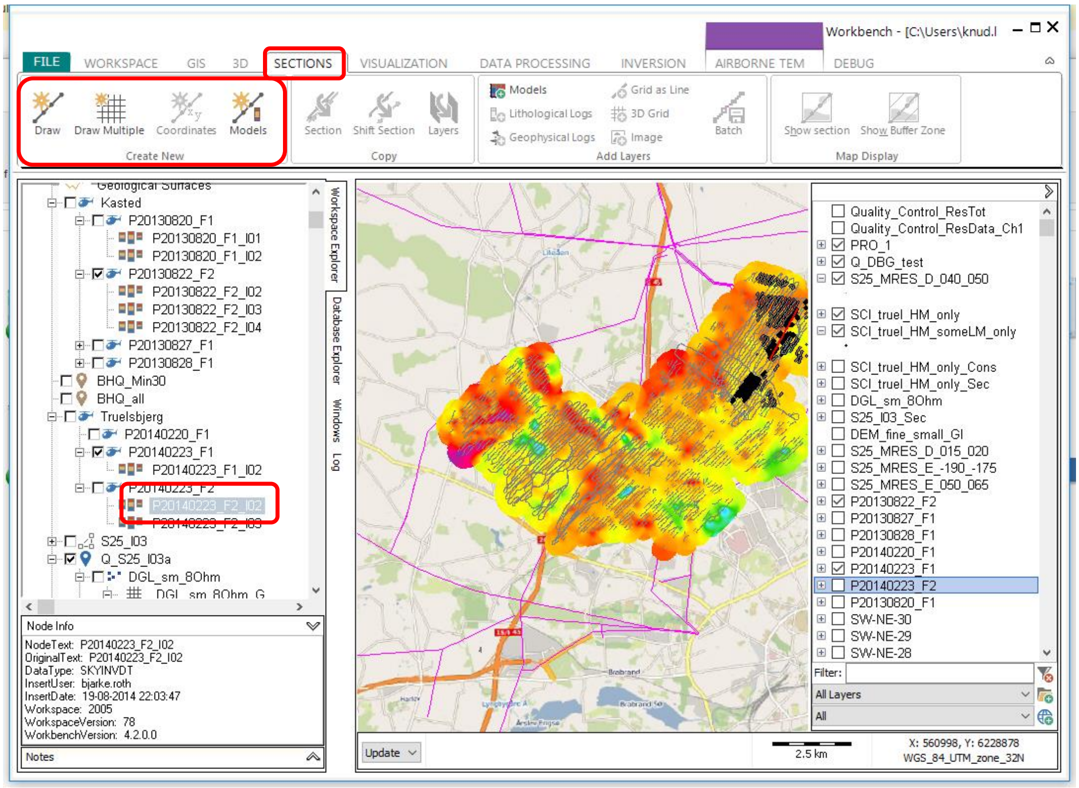

Workbench - [C:\Users\knud.lassen\Documents\test_data\skyTEM\Kasted_big]
FILE
WORKSPACE
GIS
3D
SECTIONS
VISUALIZATION
DATA PROCESSING
INVERSION
DEBUG
Pointer
Rectangle
Clear
Polyline
Polygon
Find Nearest
Select
Pan
Zoom
Undo
Center
Click
Redo
Ruler
Layer
Label
Info
Navigate
Point
Polyline
Clear
Line
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
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
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
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

Create sections and add models and data to these

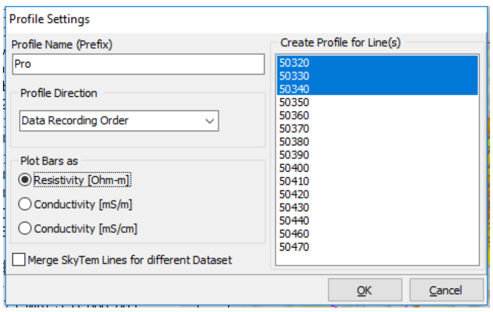
Sections are the classical way to display inversion results from different inversions together with borehole data, logs, geological surfaces etc.

Sections can be created in two ways:

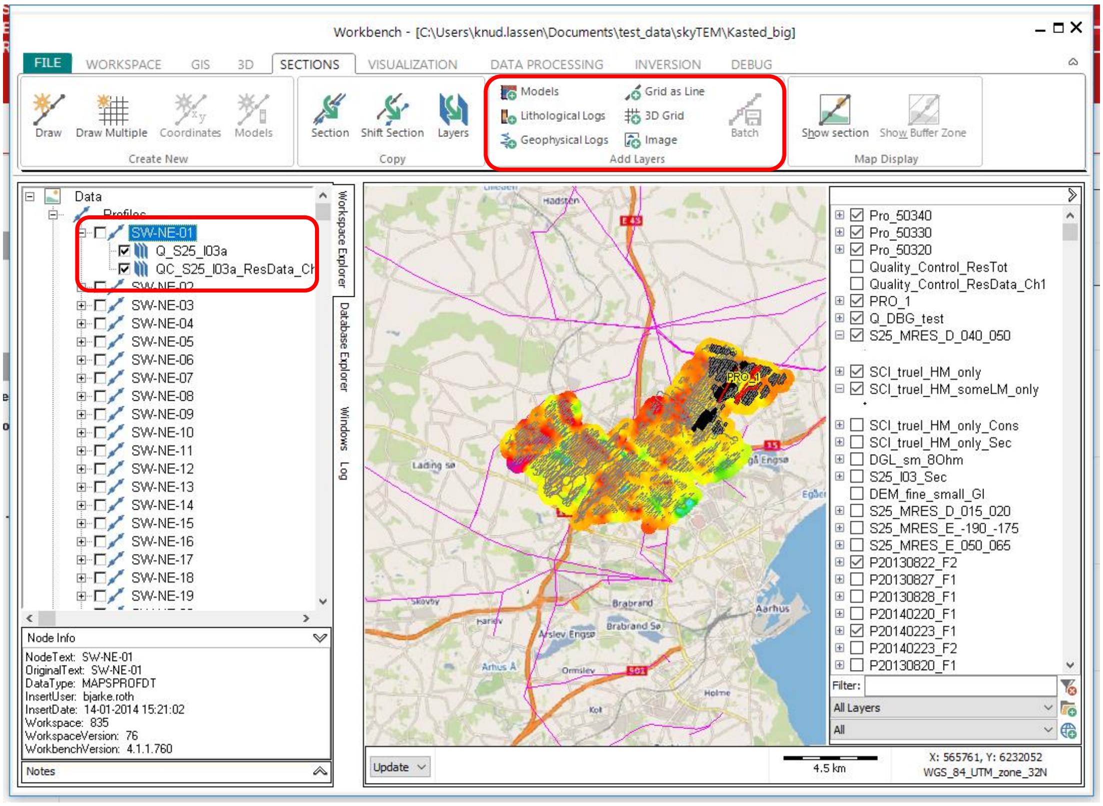
1. Under the “Sections” ribbon in the “Create new” group “Draw” can be selected, and the section can then be drawn in the GIS, single click for each bend in the section and double click to end the drawing and. After double clicking you will be prompted to name the section.
2. Draw section from inversion. 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:



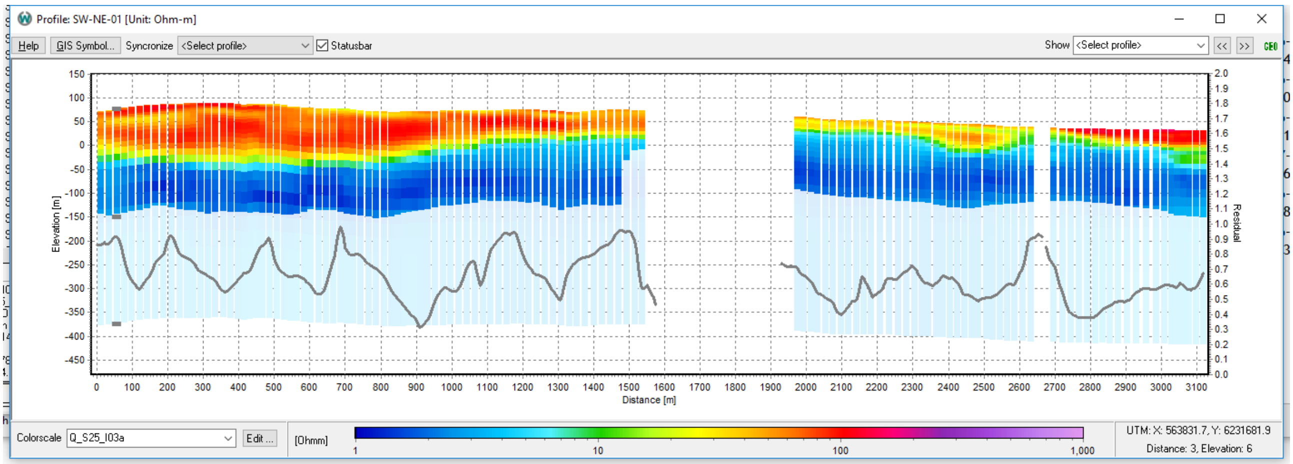
If the inversion node contains more than one data line you will be prompted to select for which lines you want to create sections, you can also select the name and how to plot the data at the section:



After pressing OK you will be prompted to select the colour scale to plot the models on, and to name the layer on the profile. Once created the sections can be found under the profile's 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:



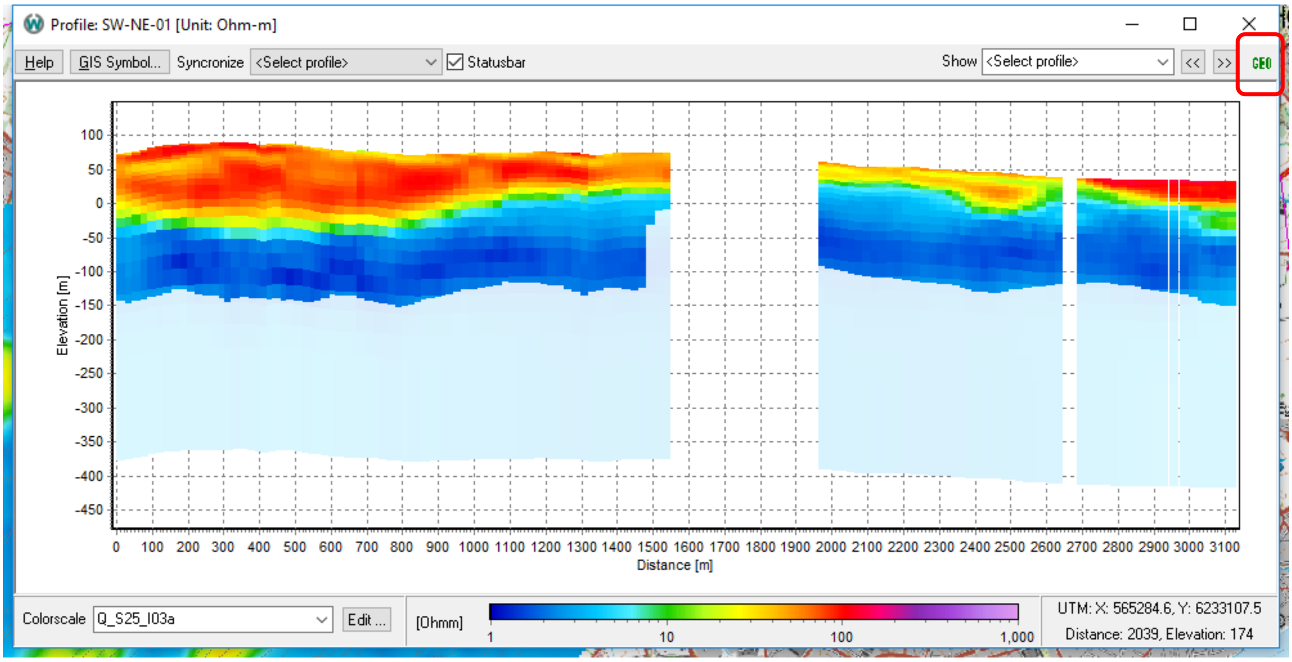
Note that the range and type (logarithmic or linear) of the axis can be edited by double clicking the axis.

Drawing and gridding geological surfaces

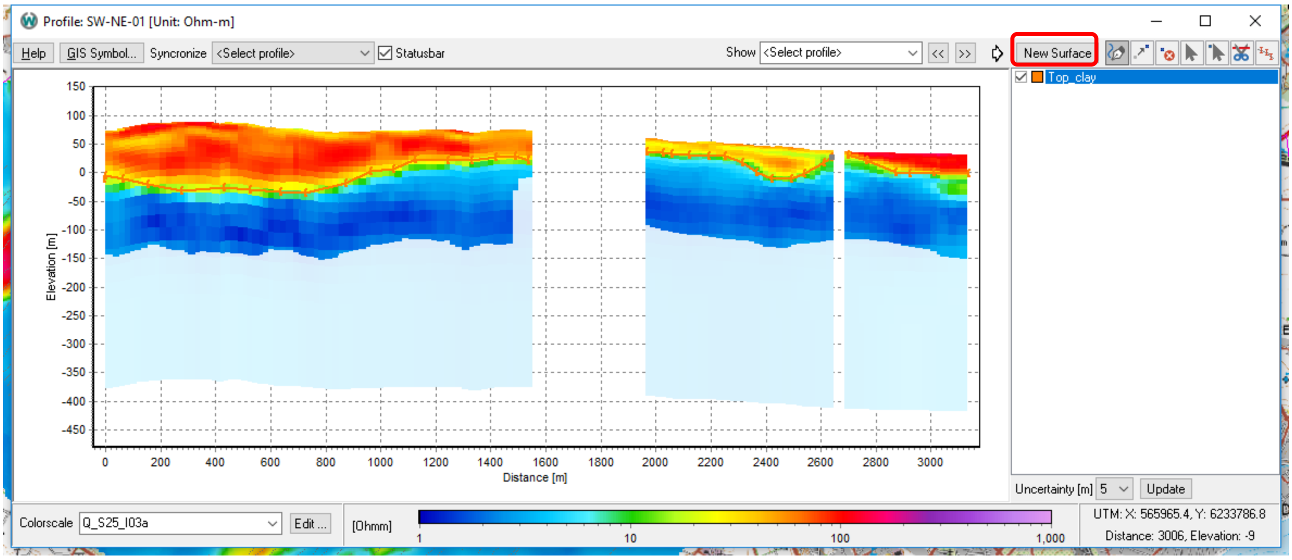
Geological surfaces are the interpretation tool of Aarhus Workbench, these can be used for tracking geological bodies and other targets, such as the interface between sand and clay, the top and bottom of a buried landfill or the water table in an aquifer, across different sections and data types.

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:



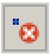
This opens a new menu next to the profile:



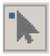
A new surface is created by pressing “New Surface” and entering a name for the surface. The surface can be edited using the controls seen below:


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

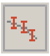

Move point, this tool makes it possible to drag and drop misplaced points


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.

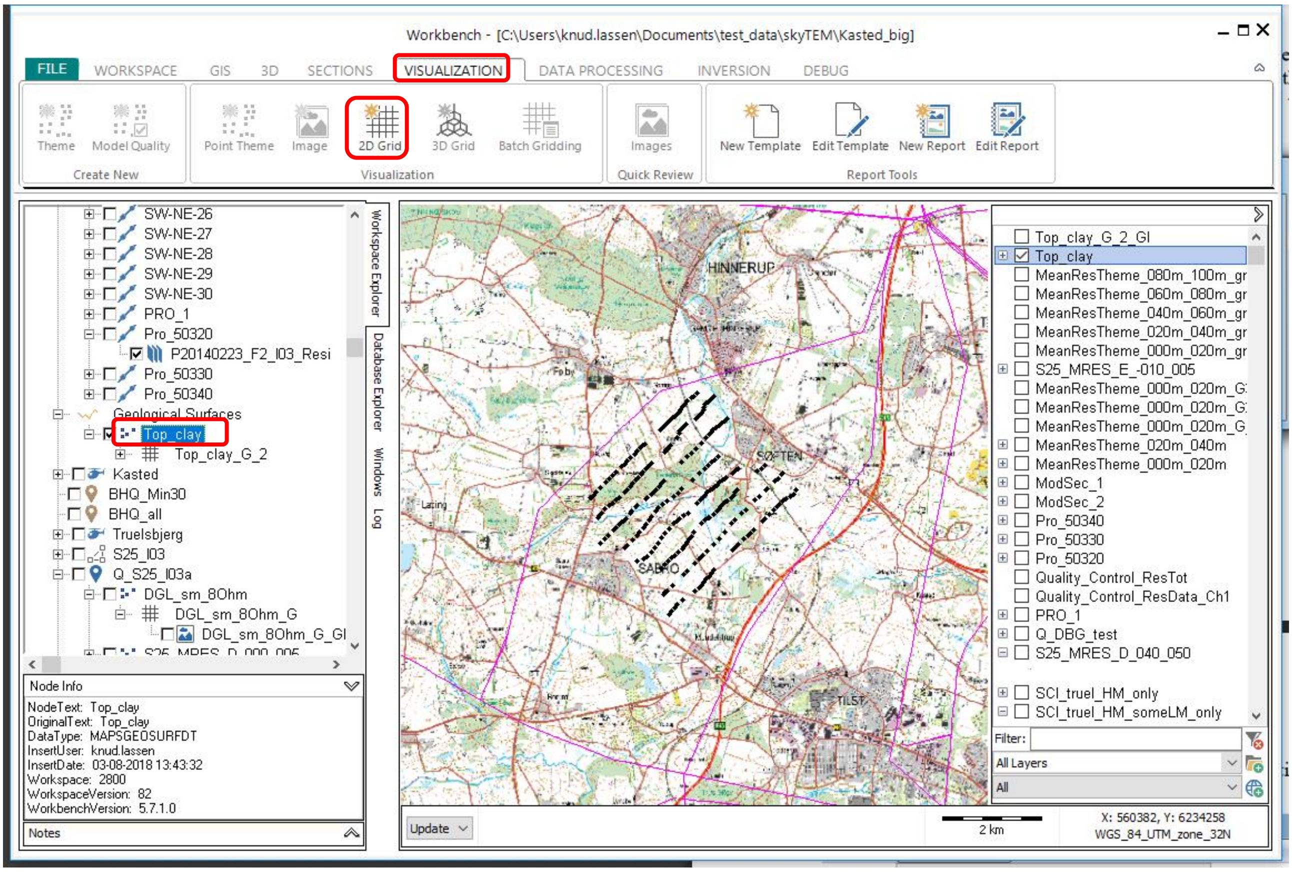
Change uncertainty, changes the uncertainty on the selected points when pressing “update” and on any new points drawn, this feature makes it possible to express the uncertainty on the exact location of the geological surface.

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

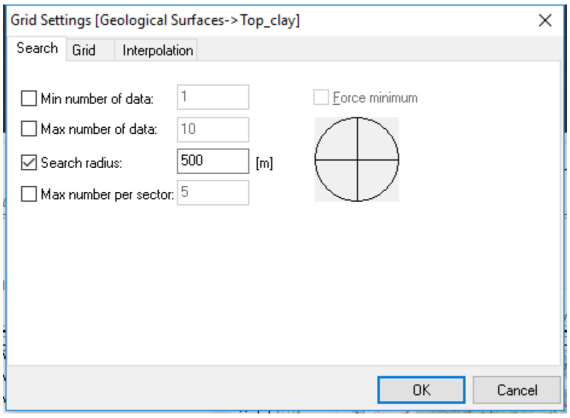
The location of the points on the geological surface can be shown on the GIS map by checking the name of the geological surface in the workspace explorer under “geological surfaces”. This can be useful in getting an overview over which surfaces are drawn for which profiles. Before this is possible it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

Gridding of geological surfaces

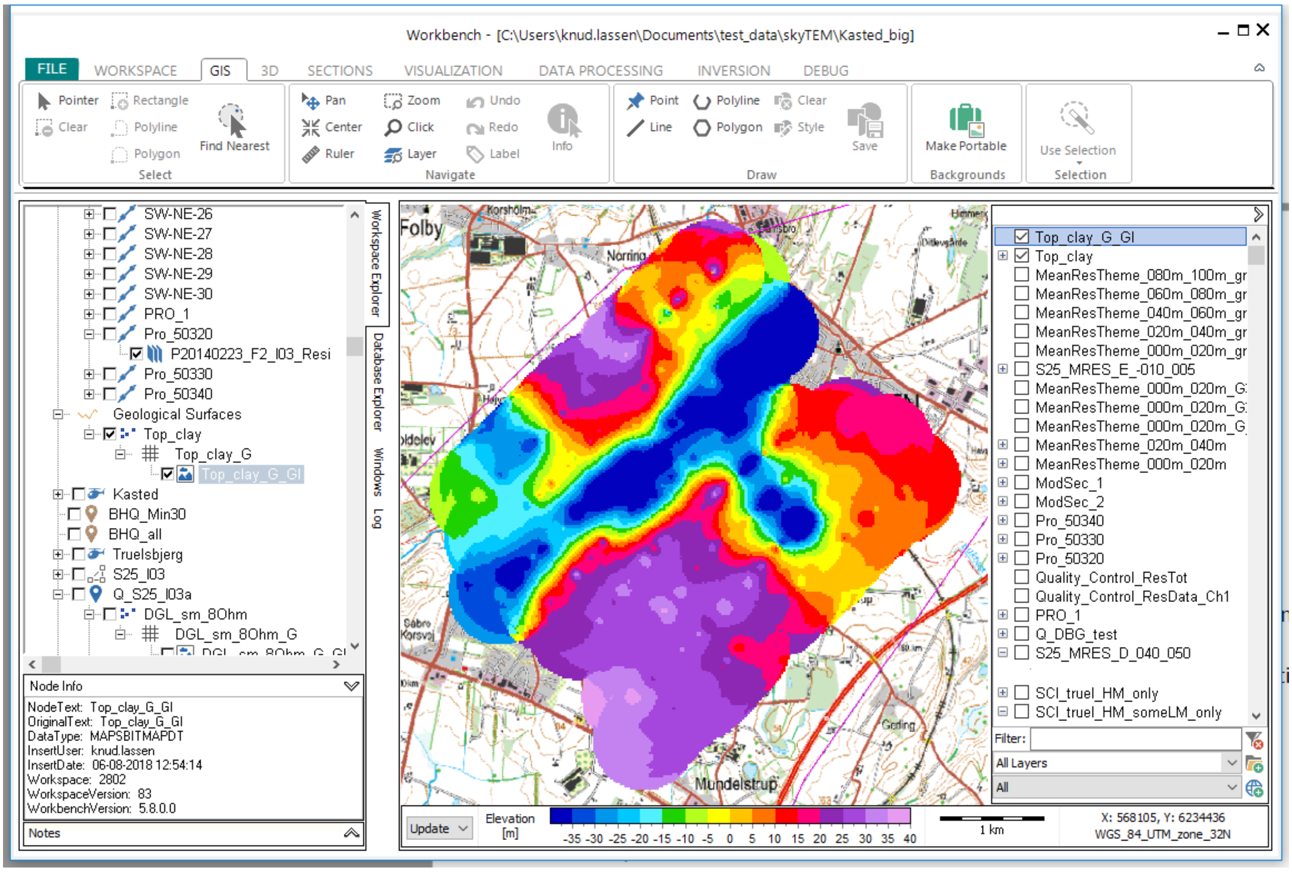
Once the geological surface is drawn on all sections/profiles in the area of interest the surface can be gridded to be shown on a horizontal map. The grid is created by navigating to the “VISUALIZATION” ribbon, highlighting the geological surface to grid in the workspace explorer and pressing “2D Grid”.



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.



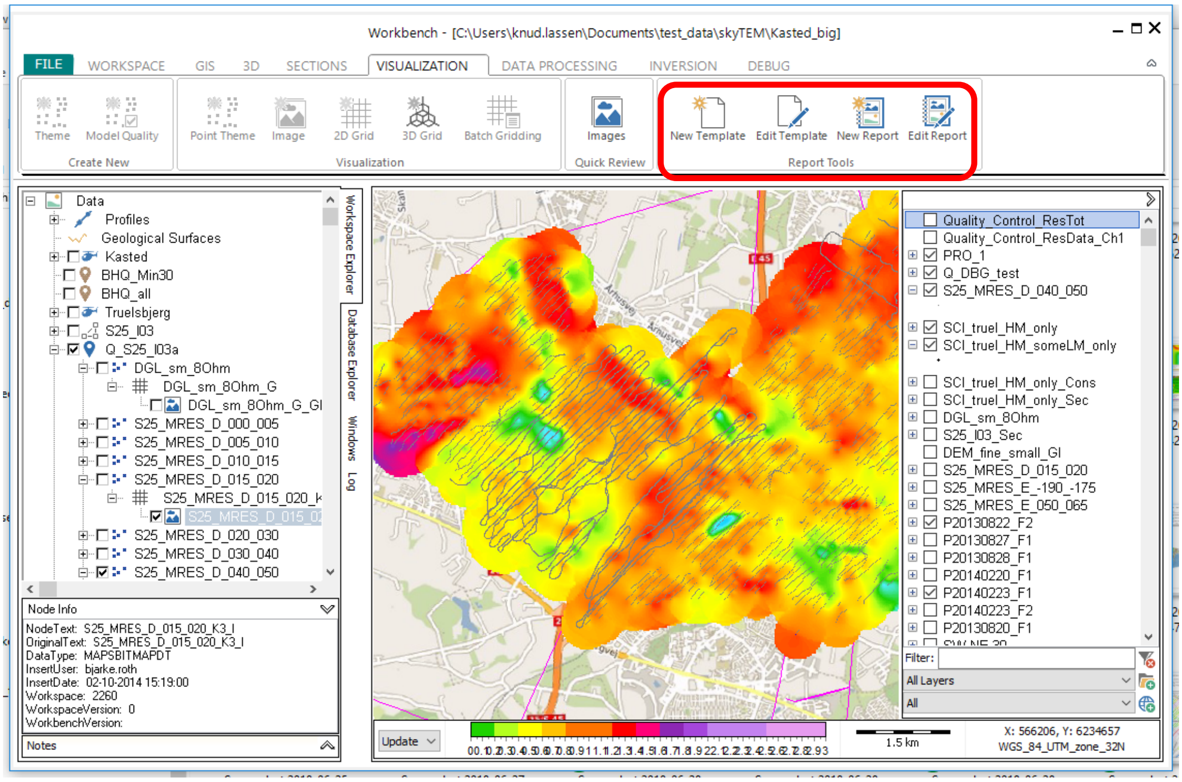
Once the grid is created it can be found in the workspace explorer under the geological surface, to display the grid on a map an image must be created for the grid. This is done by highlighting the grid in the workspace explorer and selecting "Image" under in the "VISUALIZATION" ribbon, you will then be prompted to select the colour 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
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
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_000m_020m_G
MeanResTheme_000m_020m_G
MeanResTheme_000m_020m_G
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

Creating a PDF report from template

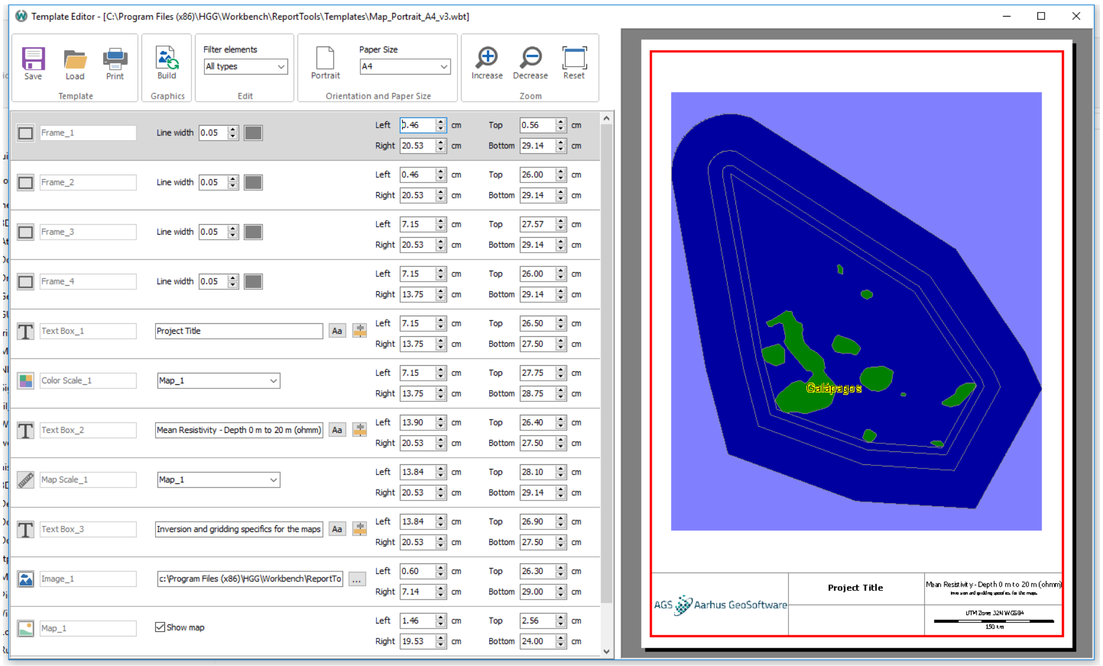
The report tool is a tool to create high resolution, professional looking reports from the different data found in the workspace, the reports can include maps, profiles, themes etc. The reports are generated from predefined templates that defines the layout of the report e.g. paper format, location of 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:



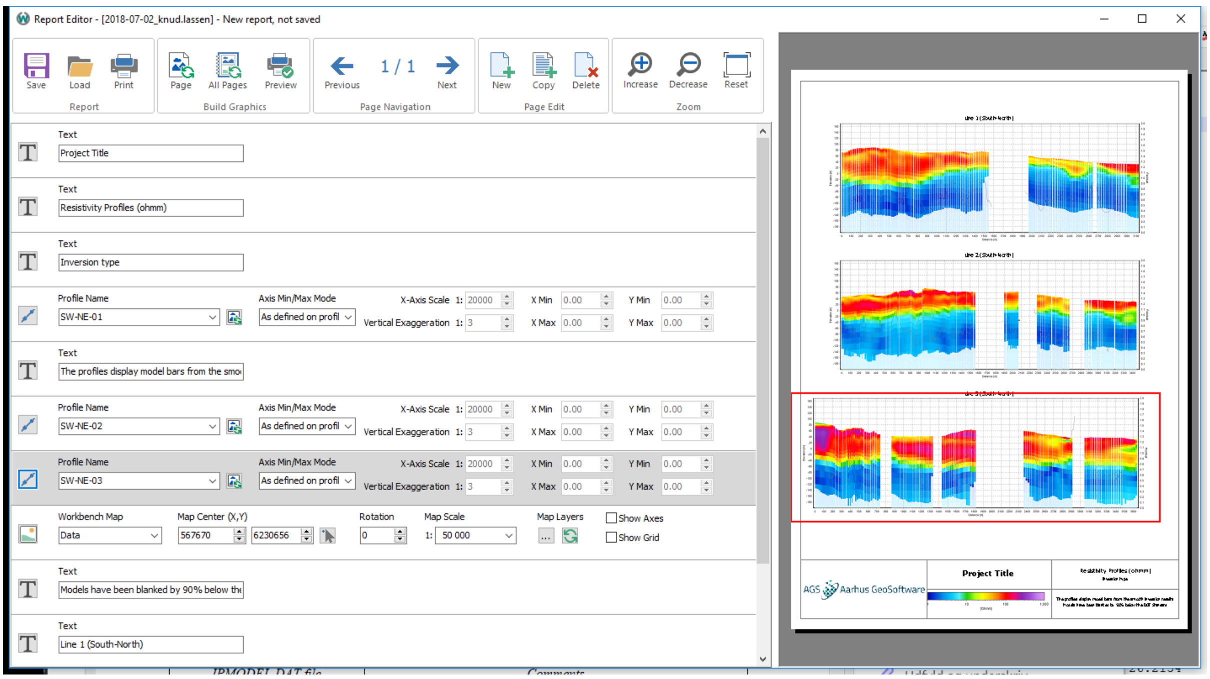
Each report page is built from the following elements: frames, text boxes, maps, legends, map scales, north arrows, images, profiles and colour scales. By selecting either new template or edit template the template editor is opened:

AGS Aarhus GeoSoftware



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.

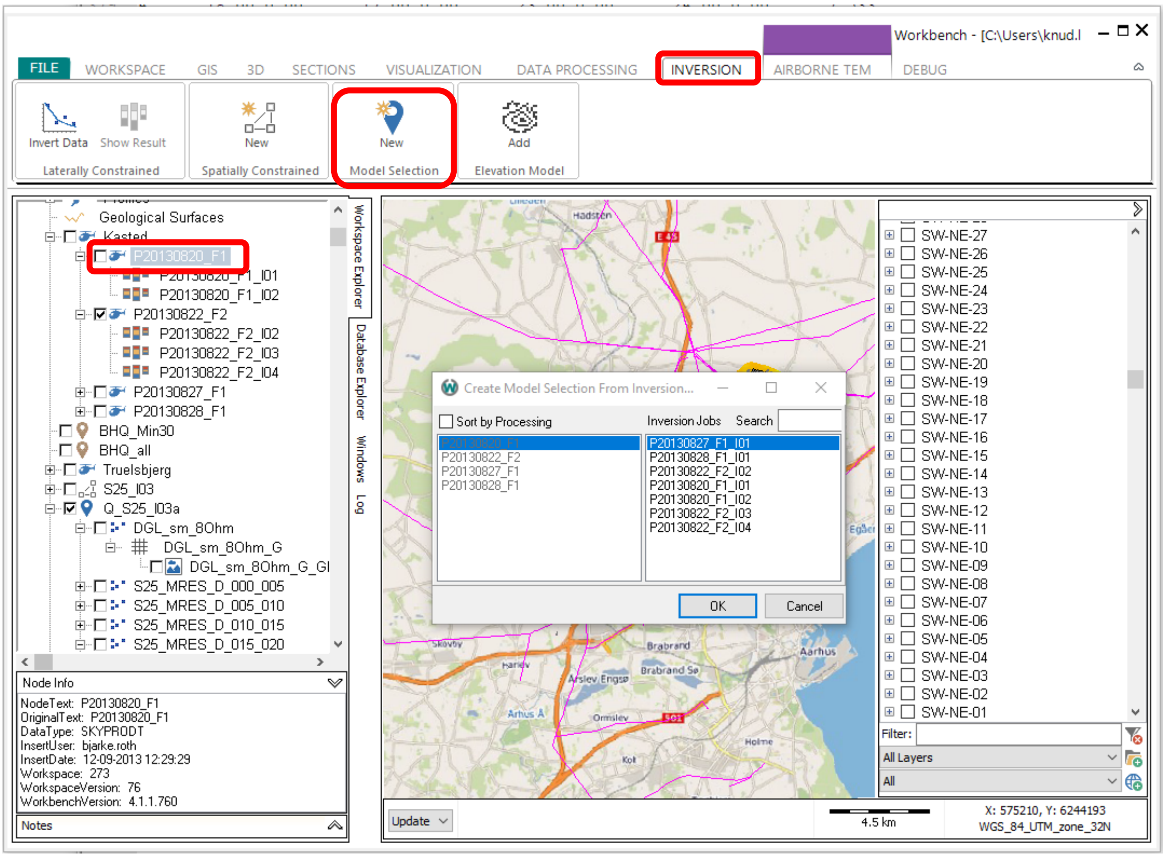


Creating and using themes

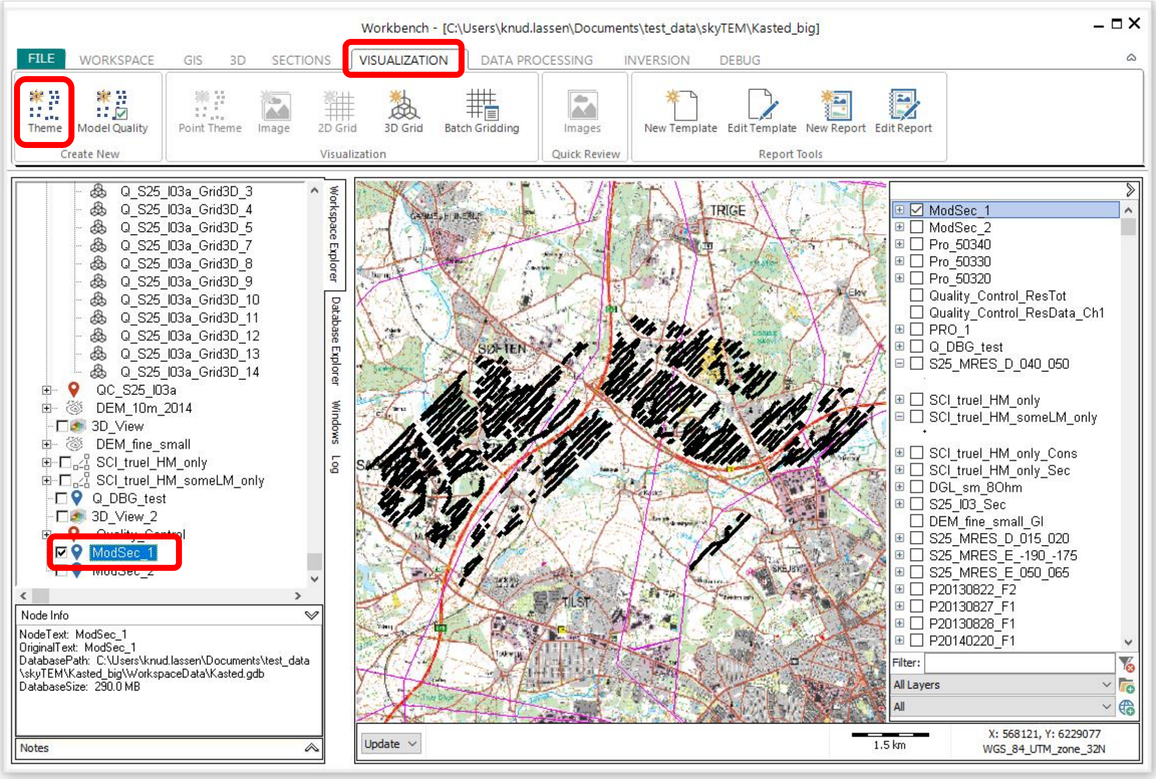
Themes are the tool for visualizing results on the map e.g. the mean resistivity of a given depth interval, the depth to a good conductor, the resistivity of the third layer in the model, or thickness of a body with a given resistivity.

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

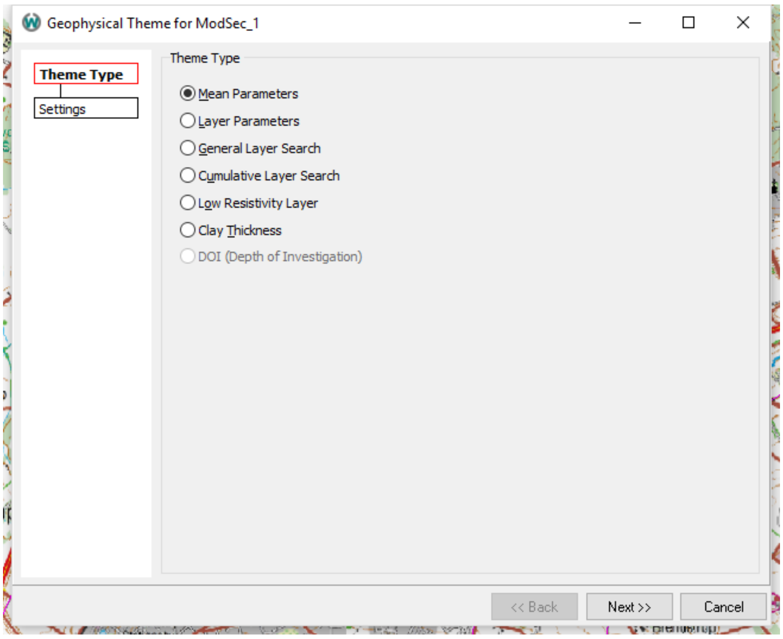
A theme is created from a model selection. A model selection is created from the “Inversion” ribbon by highlighting a processing or inversion node and selecting “New” from the “Model Selection” group. A model selection can contain models from inversions from different processing nodes by removing the 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:

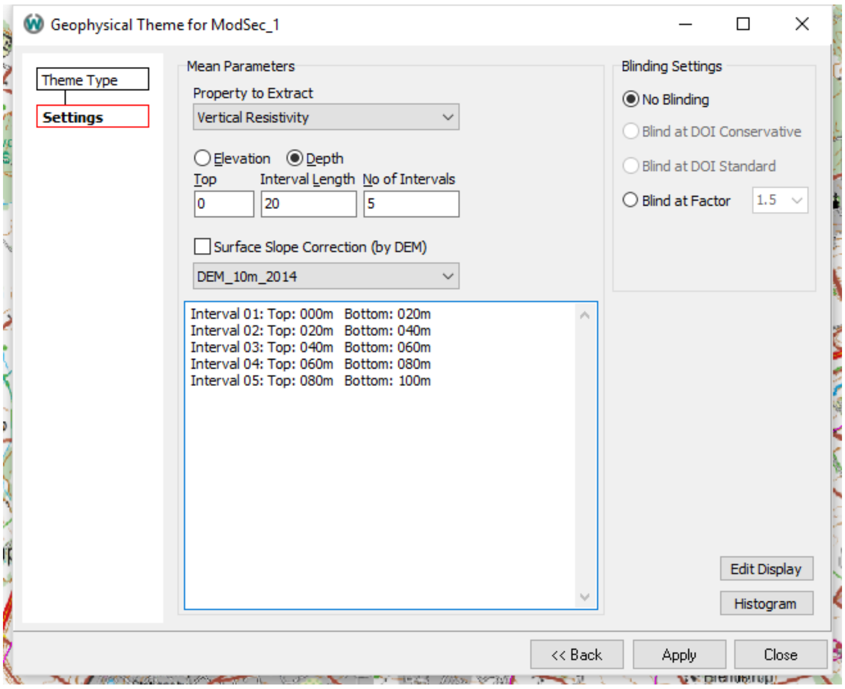


The following dialog box will then appear:



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:

AGS Aarhus GeoSoftware

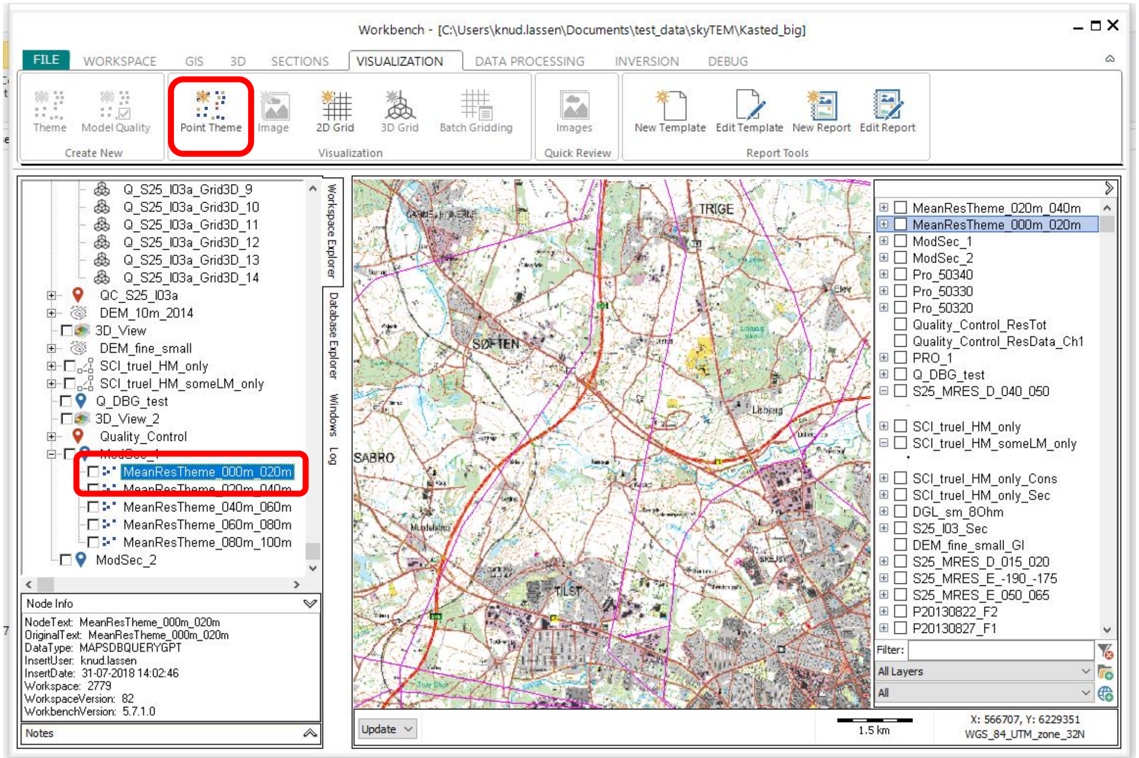


The themes are created by selecting "Apply". The themes can now be visualized as points or 2D grids.

Visualizing themes

Points

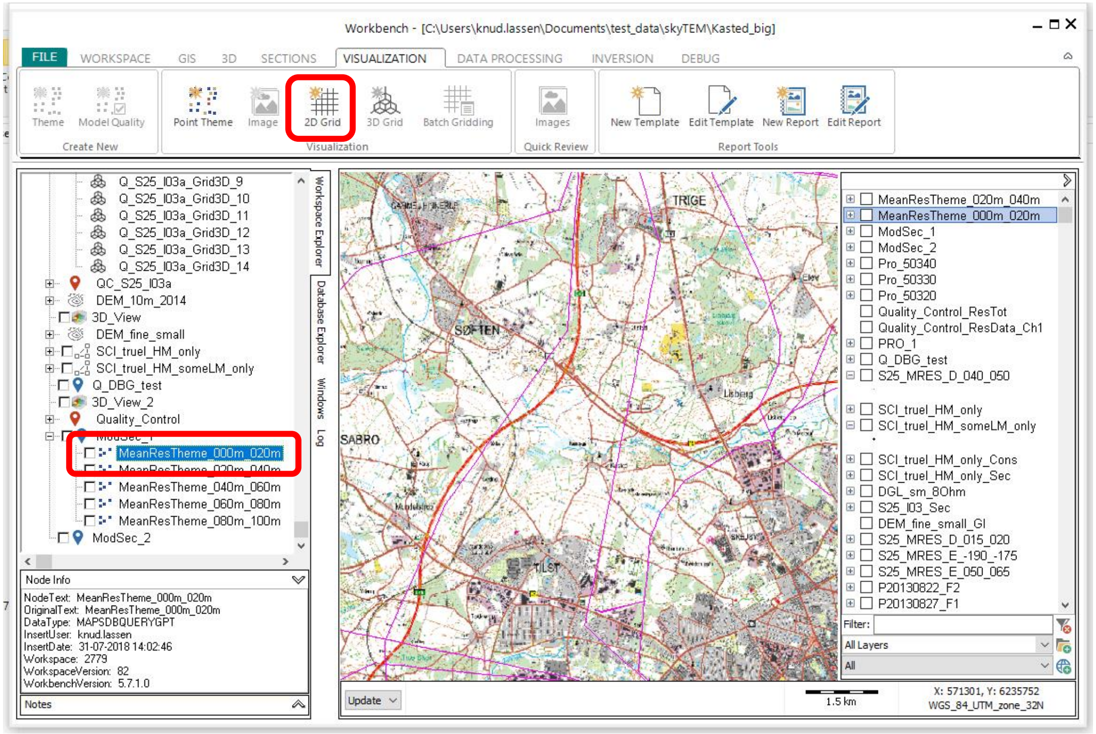
To plot the gridded values as coloured points in the GIS window highlight the theme and press "Point Theme"



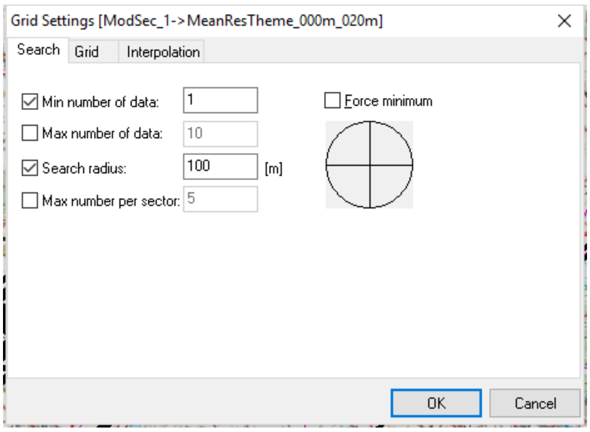
Select the colour scale, point size and point shape in the dialog box and press “OK” and select the name of the theme, the point theme can now be found in the workspace explorer.

2D Grids

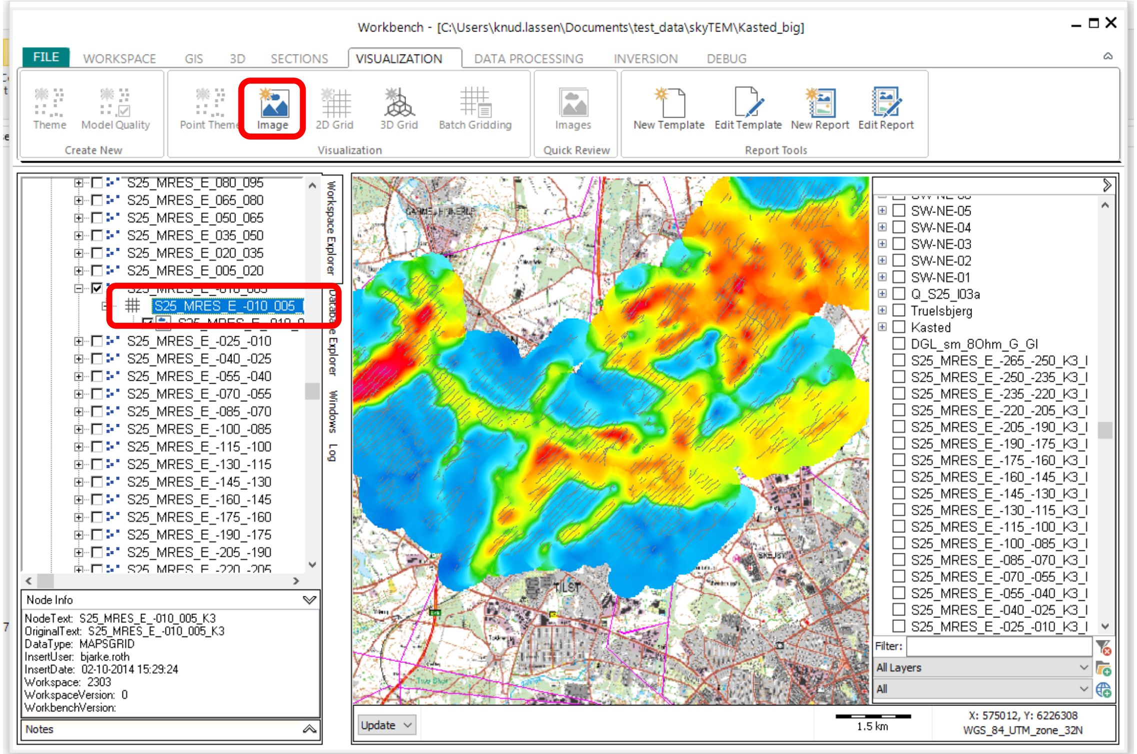
The other way of presenting themes are as surface covering interpolated grids, these are created by highlighting the theme and pressing “2D Grid”



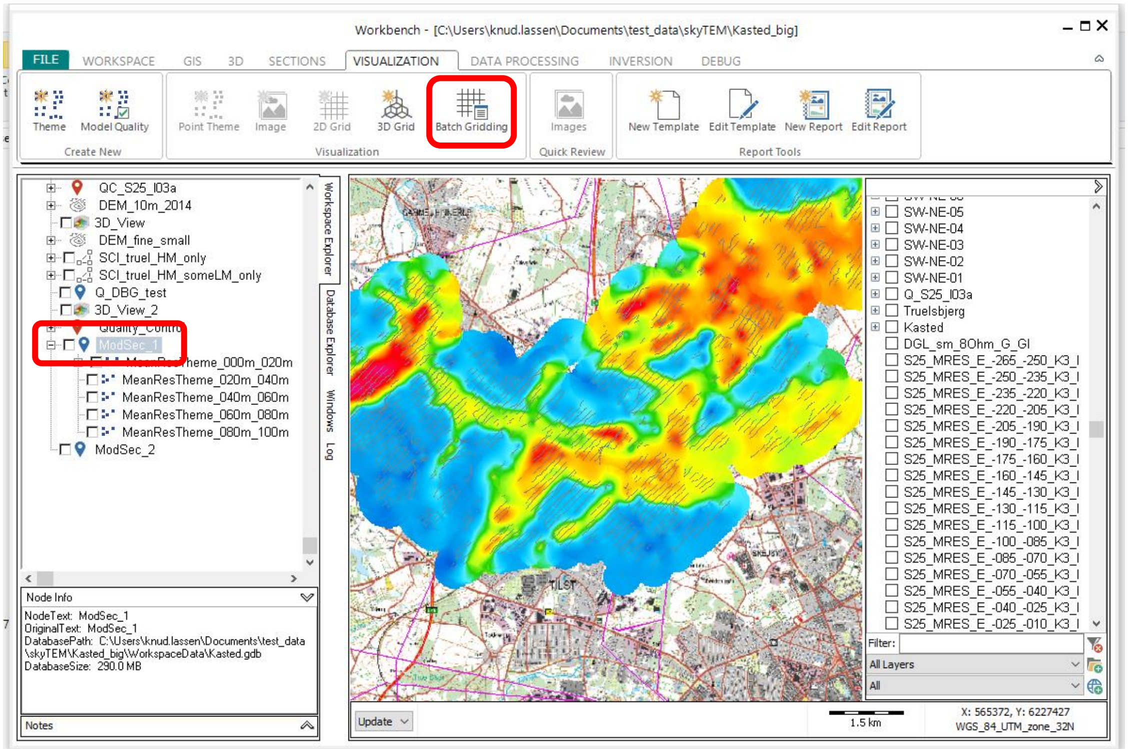
The following menu box will appear:



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 colour 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 colour 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”:



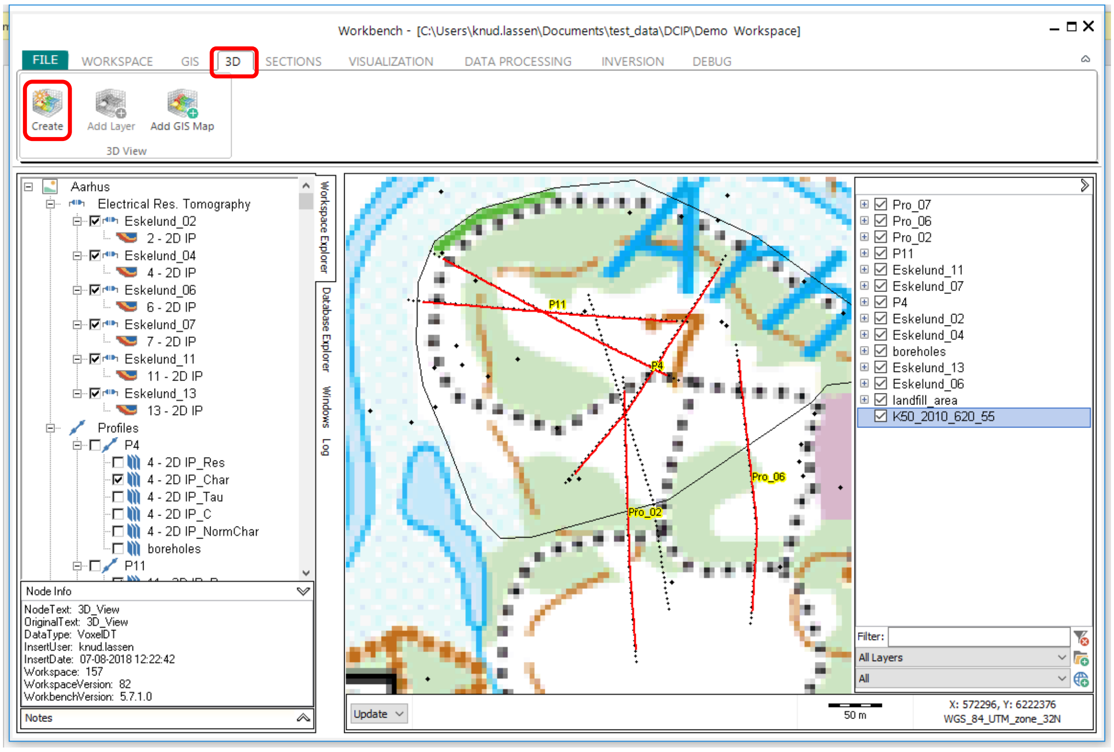
Using the 3D viewer

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

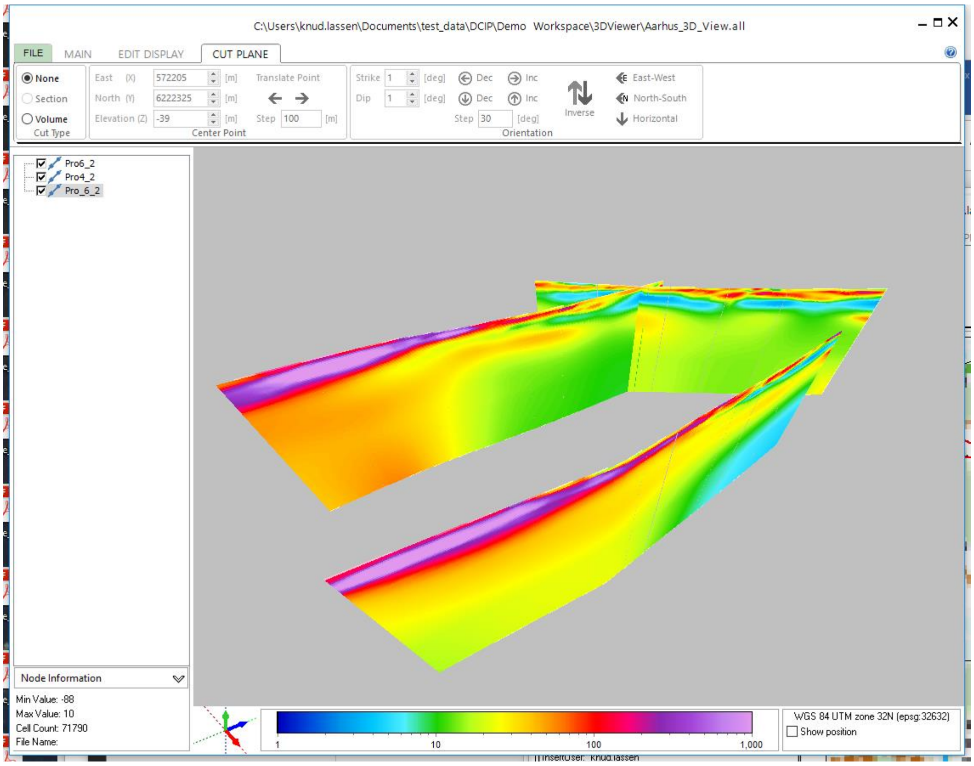
To create a new 3D view, navigate to the “3D” ribbon and select “Create”:



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 colour 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:



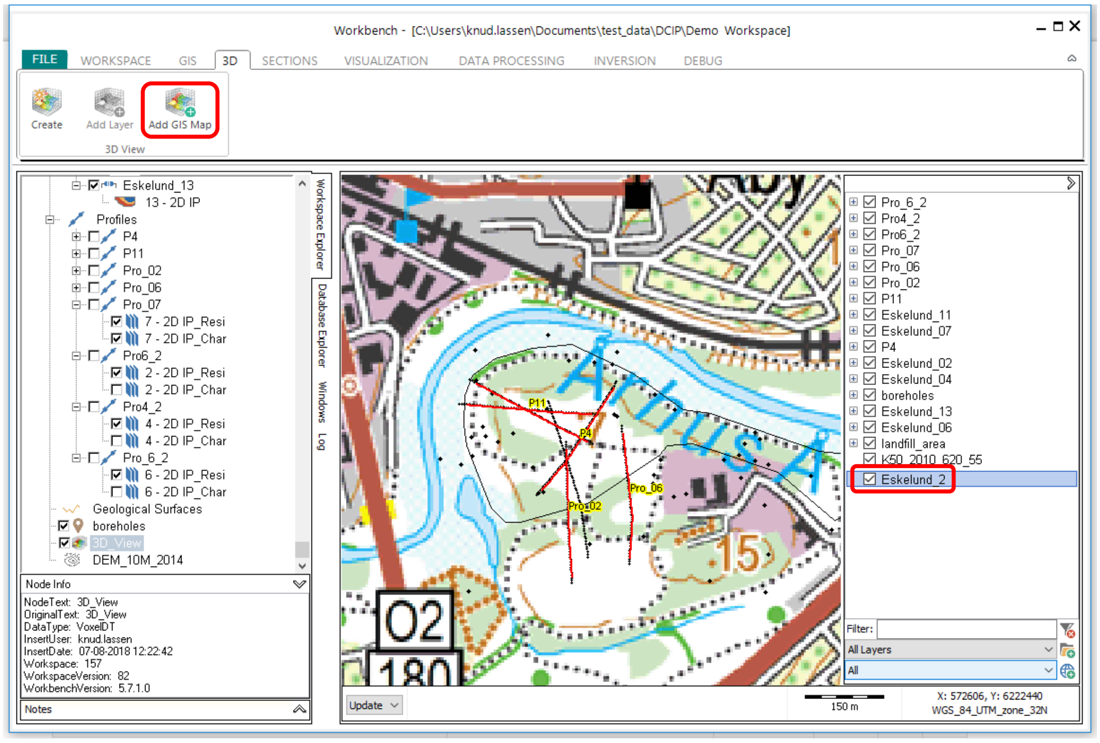
Navigating in the 3D viewer

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 centre of rotation by holding down the right mouse button and dragging the content, the centre of rotation can be shown by checking the “centre” 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:

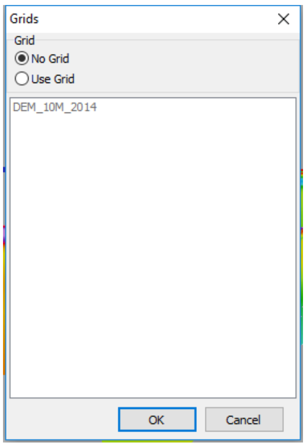


Location referenced images can also be imported directly to the 3D viewer by pressing “File->Load file”.

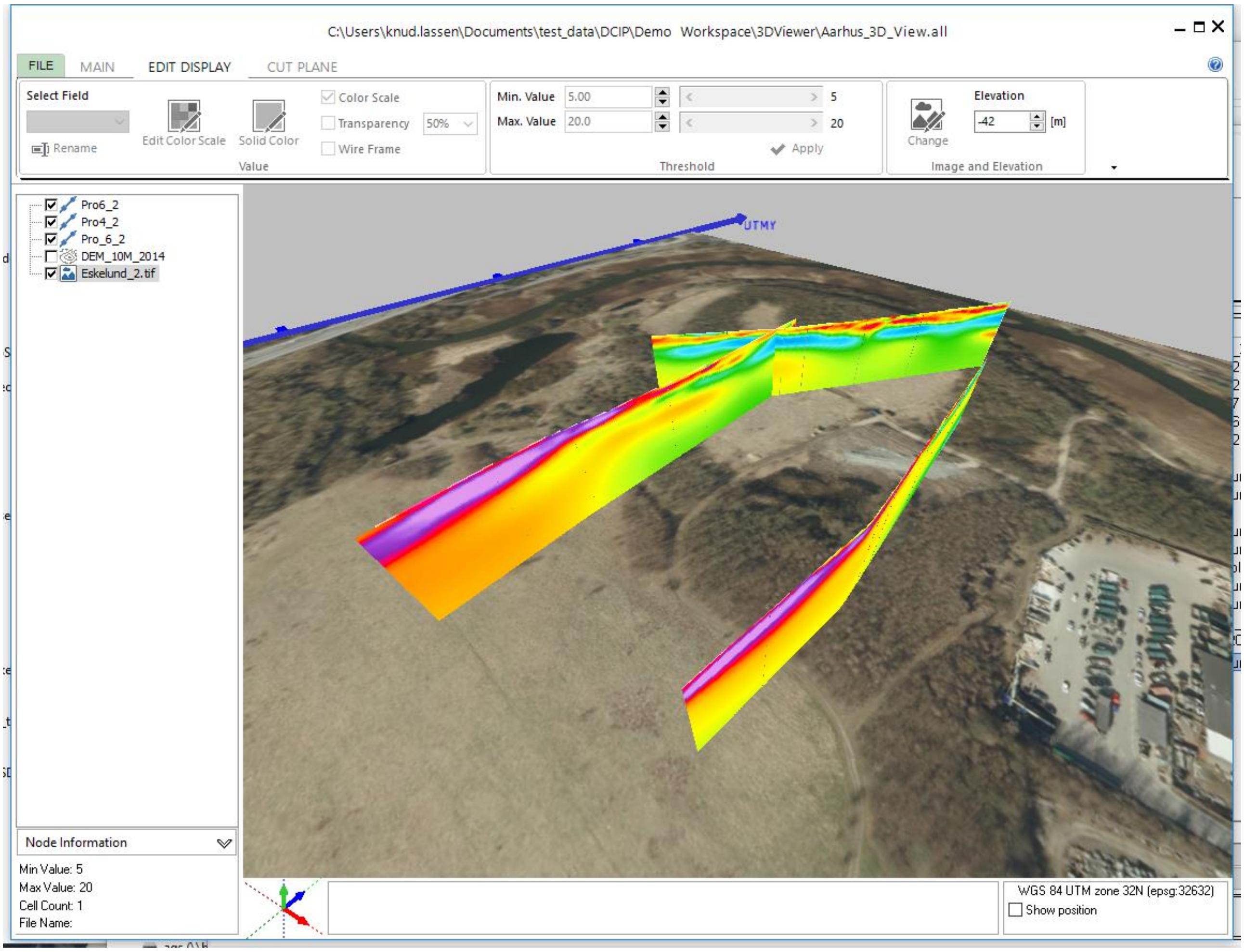
The other type of images is those created from gridded geophysical themes or gridded geological surfaces, these can be highlighted in the workspace explorer and added to the 3D view in the same ways as sections.

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:

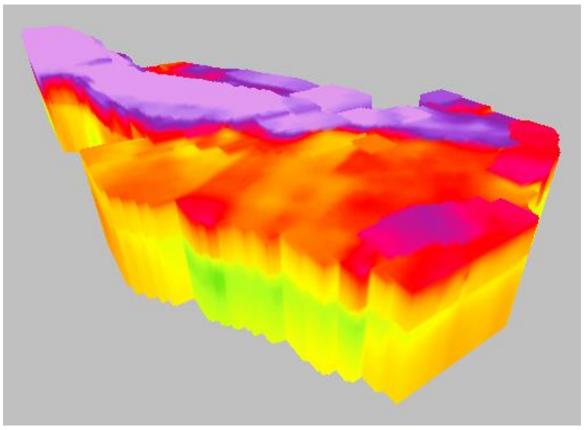


Cut planes, thresholding and 3D grids

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

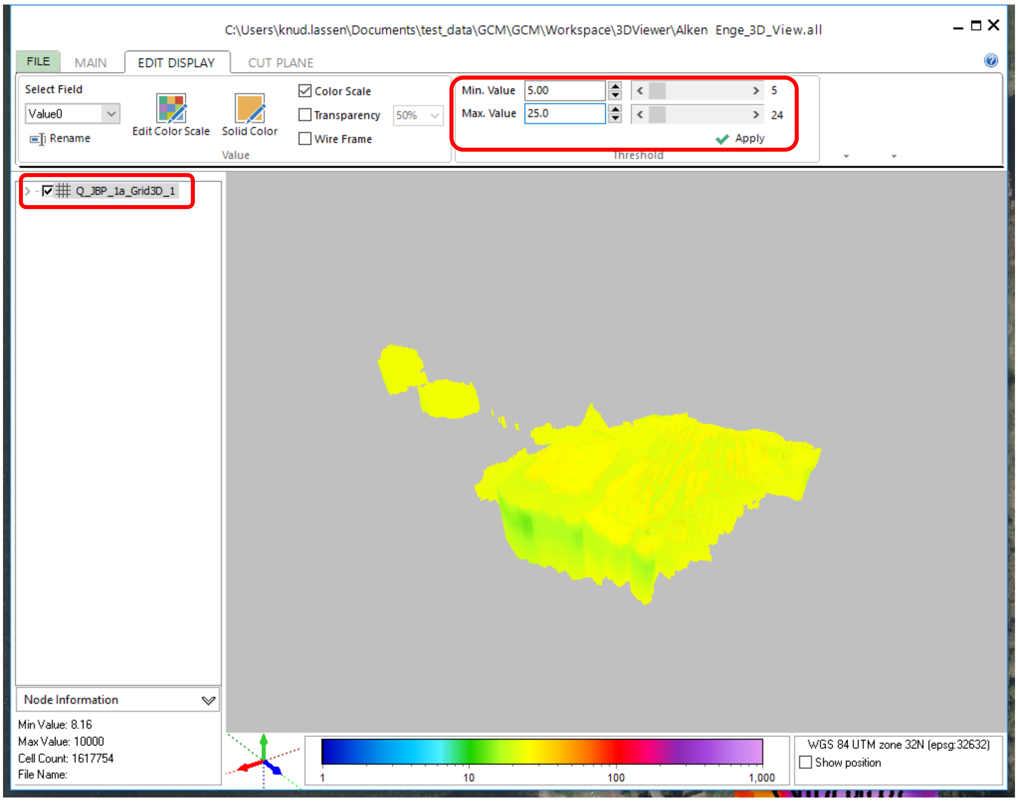
3D grids are added as all other data types by highlighting in the workspace manager and selecting "Add Layer", they can also be loaded in directly as VTK files e.g. from external inversion or modelling software e.g. Res3DInv.

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:

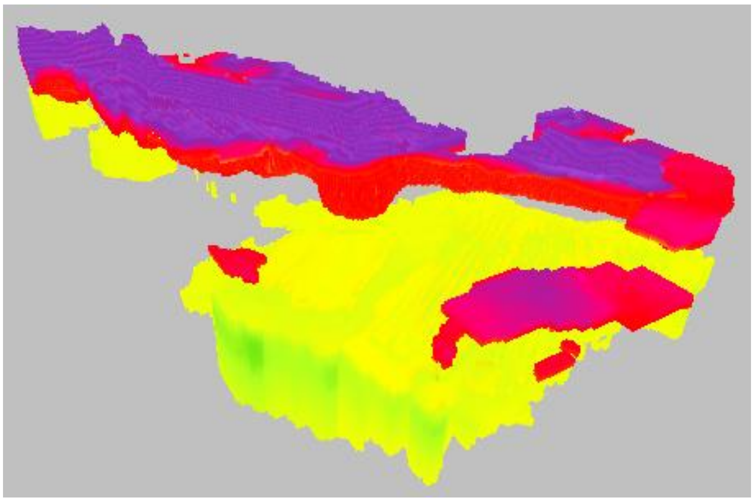


Thresholding

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”:

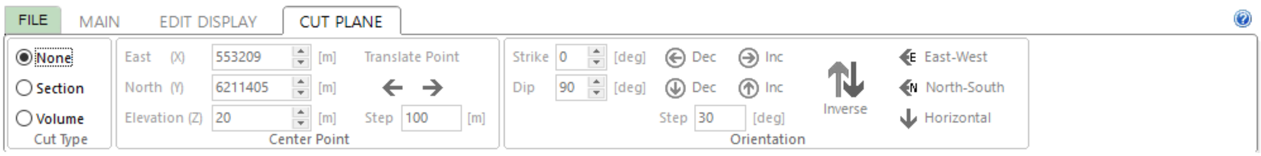


It is possible to show more than a single range of resistivities at a time, this is done by adding subnodes to the object. This is done by highlighting the object and selecting “Add Subnode” in the “Main” ribbon. Each of these subnodes can then be given different threshold values as shown below:

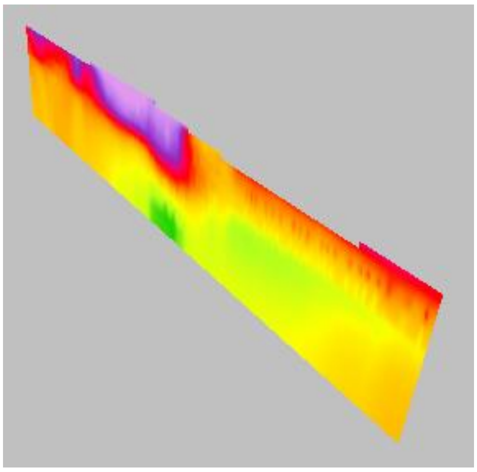


Cut planes

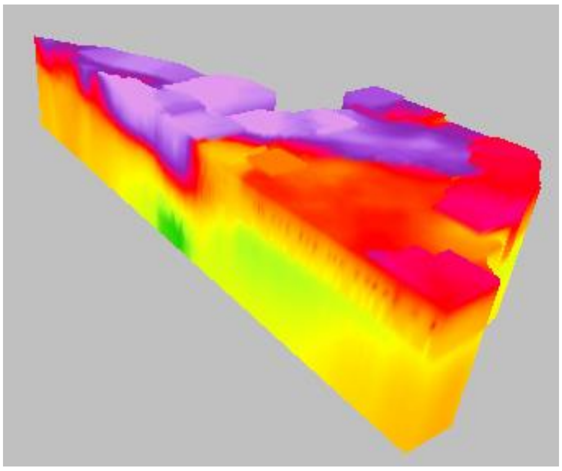
Cut planes makes it possible to make cuts through data to show relevant regions, they are found on the "CUT PLANE" ribbon:



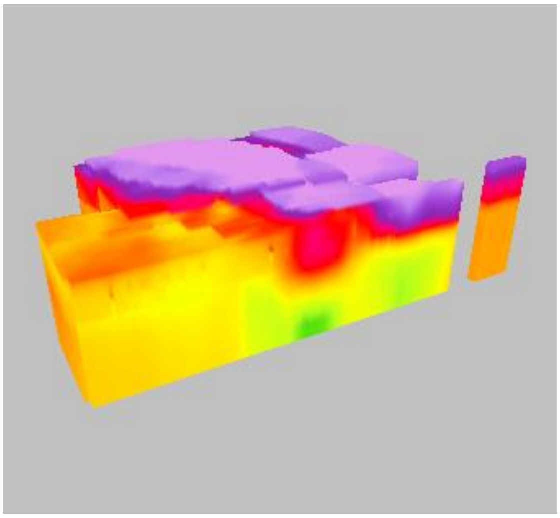
The default setting is "None", there are two additional settings, namely "Section", showing a slice through data:

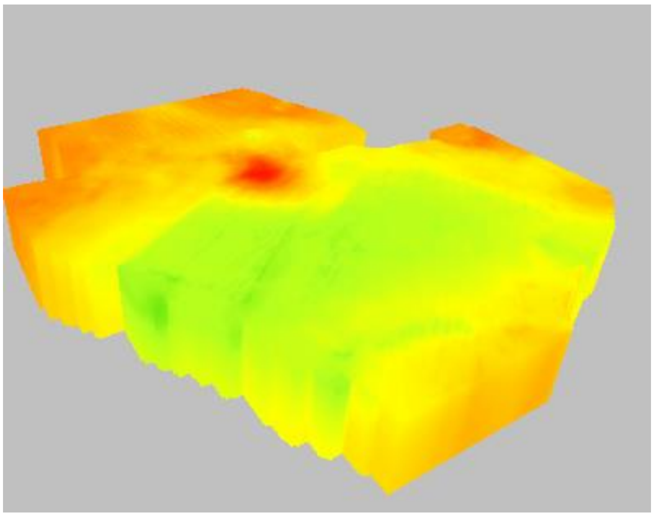


And "Volume" that cuts out part of the data:

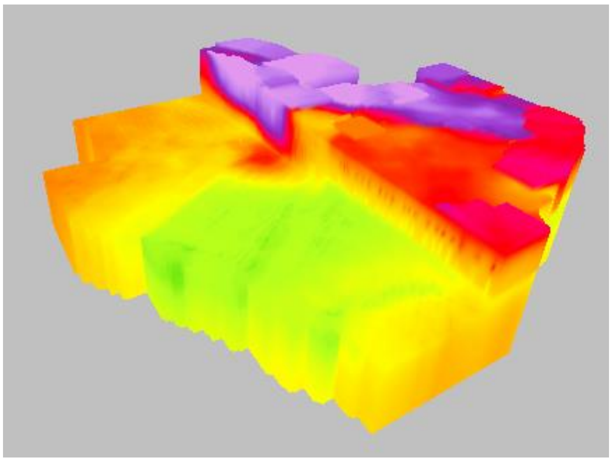


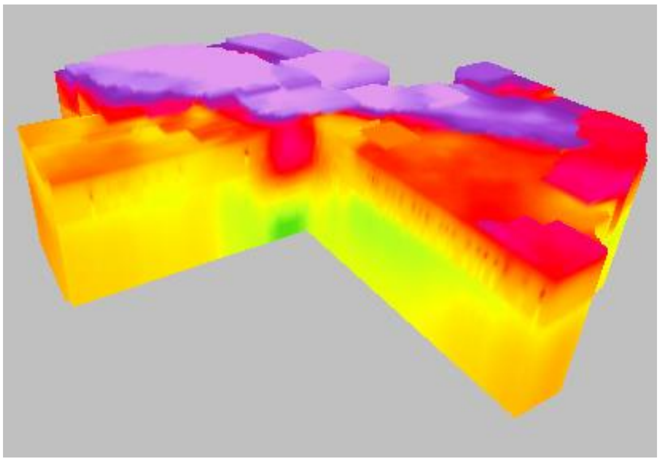
It is possible to change the location and orientation of the cut plane using the controls in the “Center Point” and “Orientation” groups:





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.

AGS Aarhus GeoSoftware

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.

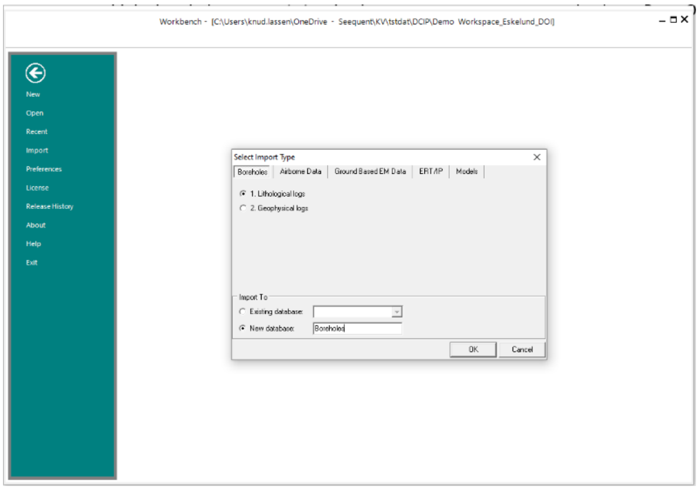
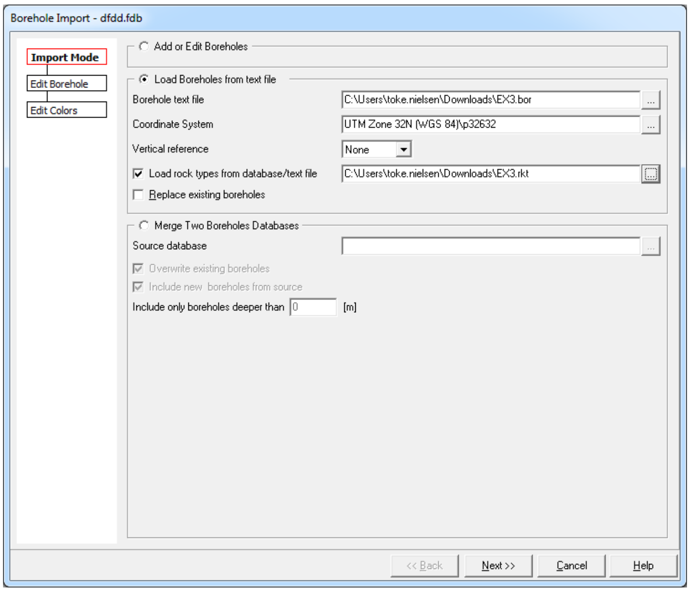
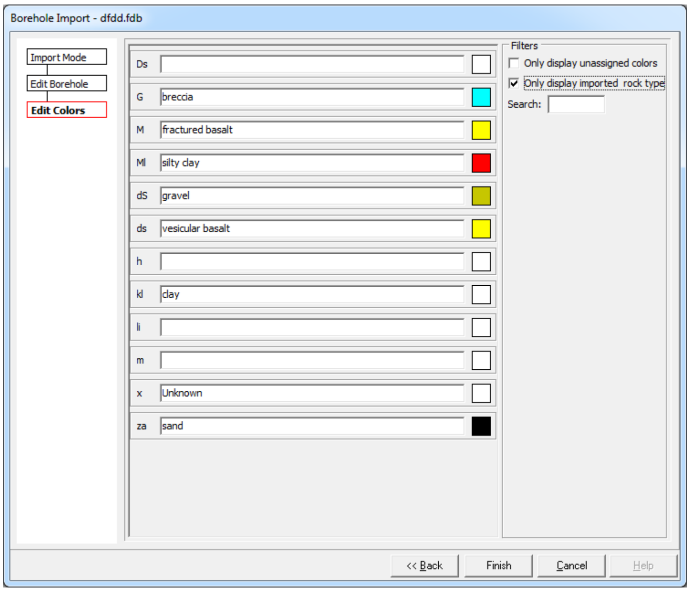


Figure 6

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”.



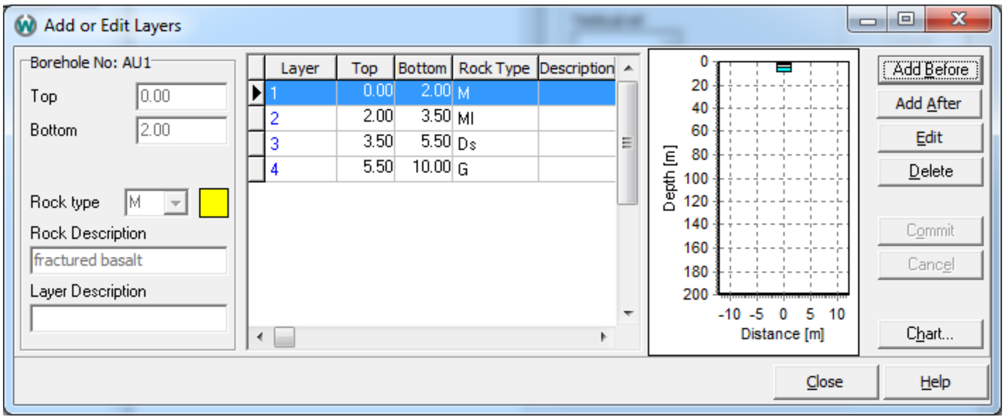
If the .bor file used does not contain the columns RGB and RockDescription, after pressing “Next”, no description and the same white color are associated to each rock type (Figure 3). It is possible to use the editing window in Figure 3 to manually modify the color (by left click on the squares on the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



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 - Centre
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%)