

Workflow for import, processing, inversion and presentation of ERT data in Aarhus Workbench

This guide presents the common workflow options, from workspace creation to final presentation of data, for ERT/DC resistivity 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.

Create the workspace p. 3

The workspace is the main tool for managing a project in Aarhus Workbench, the entire process from data import to the final visualization and presentation is contained in the workspace.

Data import p. 5

It is now time to import the ERT data from the instrument into the workbench.

Data processing and inversion p. 8

Data processing is the task of removing couplings and bad data points, so that the final inversions only contain desired information about the subsurface, unpolluted by bad data acquisition and couplings to manmade structures. Processing and inversion is an iterative process where data are processed and inverted, reprocessed and reinverted in several iterations.

After a satisfactory inversion result is obtained several options for presentation and interpretation exists.

Creating themes for quality control p. 16

Plotting maps of different inversion properties can help assess the quality of the produced inversions.

Creating sections p. 19

Sections are lines drawn in the GIS interface to which all kind of information, including inversion results, borehole information, borehole logs and geological surfaces can be added.

Creating themes for visualizing and presenting results p. 22

While sections are used to present data in vertical slices, themes present data in the horizontal plane on a map e.g. through mean resistivity maps for a given depth or elevation interval.

Adding a background map p. 14

It is often useful to have some kind of background map to make it easier to relate data and model locations to the real world, and to aid processing. It is both possible to add local maps in different formats, and to use online map services (WMS layers).

Drawing and gridding geological surfaces p. 27

Geological surfaces are the interpretation tool of Aarhus Workbench, used for tracking geological bodies and other targets across sections and data types.

Creating PDF reports p. 31

The report tool is a handy tool for presenting and delivering results to customers, once a template is created it is easy to export new data and inversion result in a consistent and professional way.

Creating 3D views p. 33

3D views are rotatable 3D models to which inversion results, boreholes, sections, maps and aerial photos can be added for impressive and intuitive presentation of results.

Import borehole information p. 41

Information from boreholes can be an extremely useful tool in verifying inversion results and for aiding geological interpretation.

Import geophysical logs p. 44

Information from geophysical logs can also be of great help in the interpretation phase.

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.

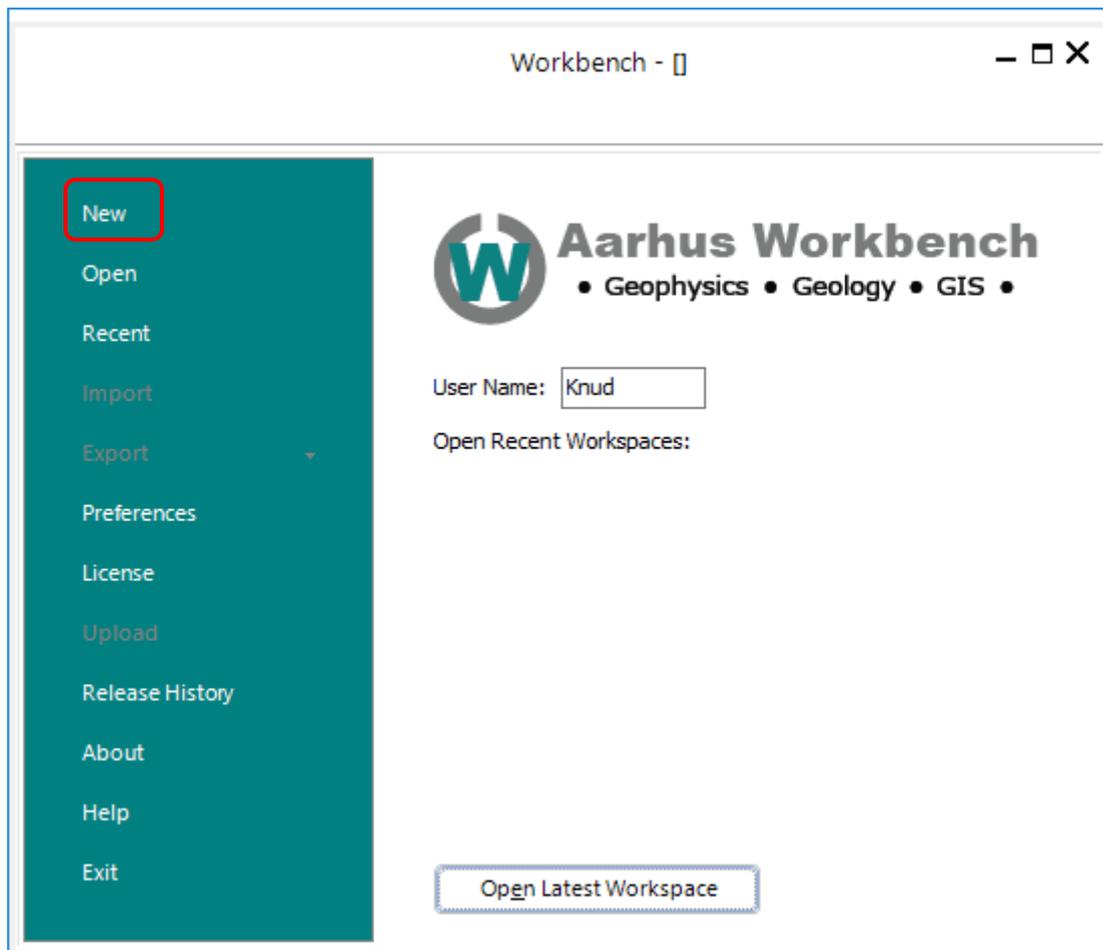


Figure 1 - Creating a new workspace

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.

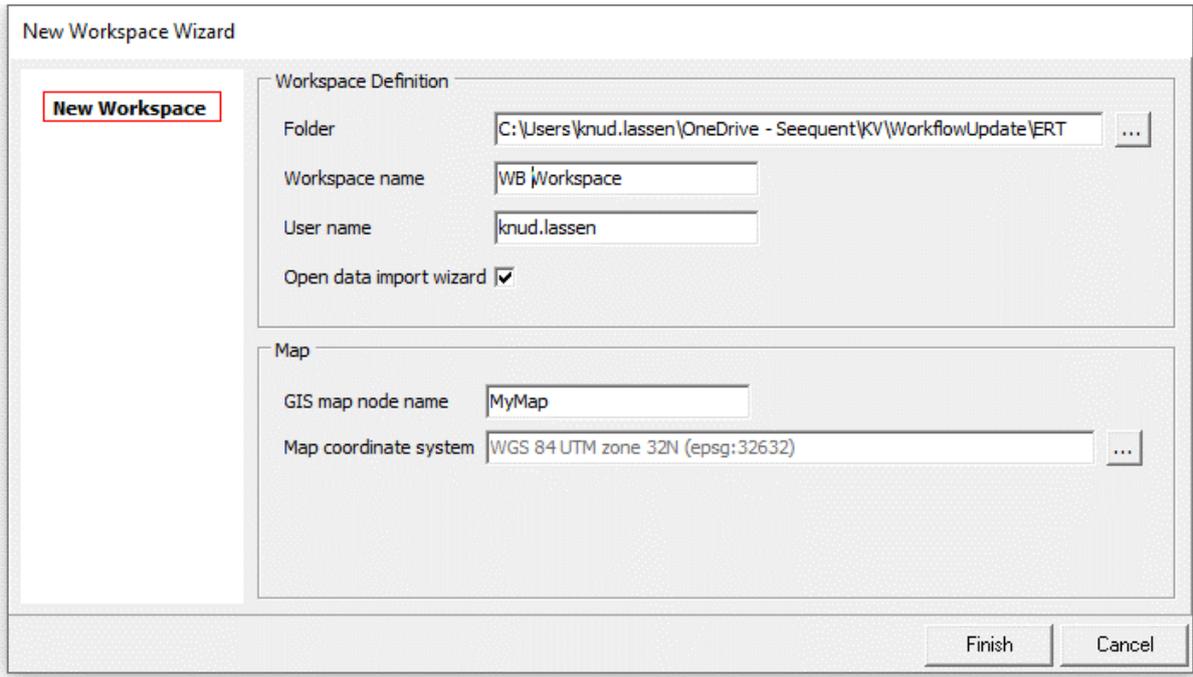


Figure 2 - Setting up the workspace

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.

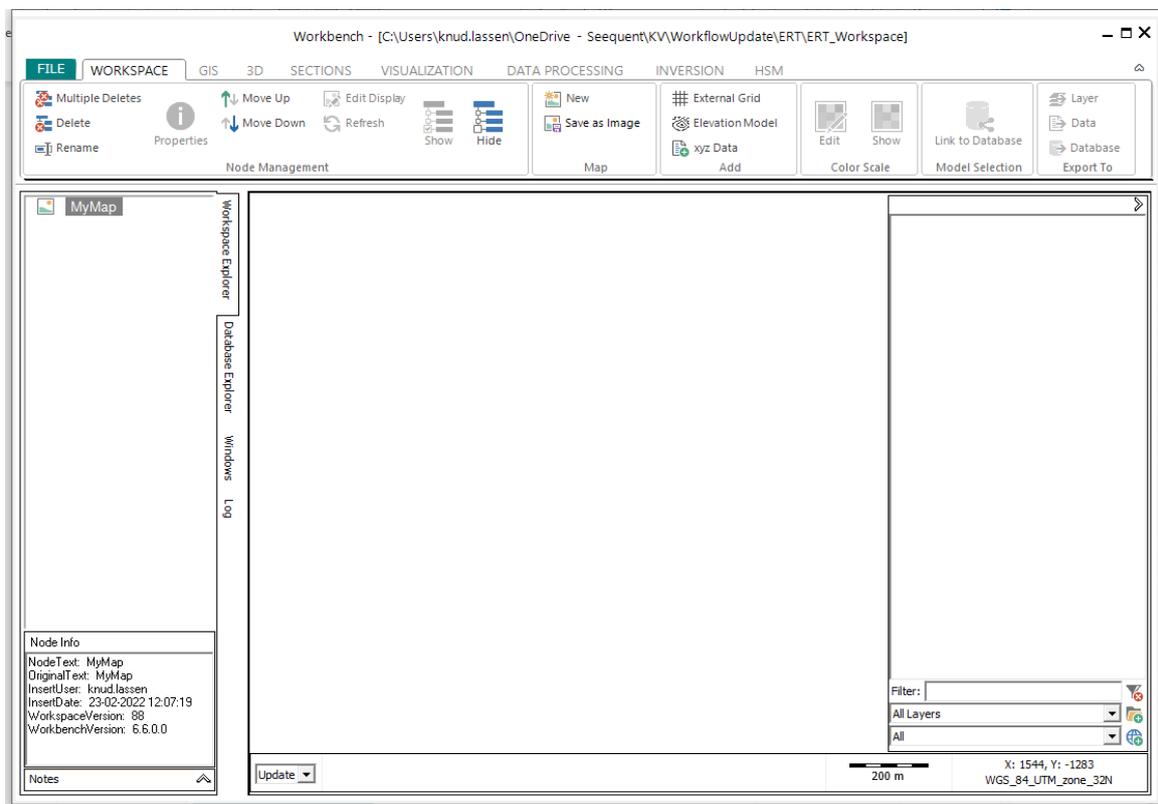
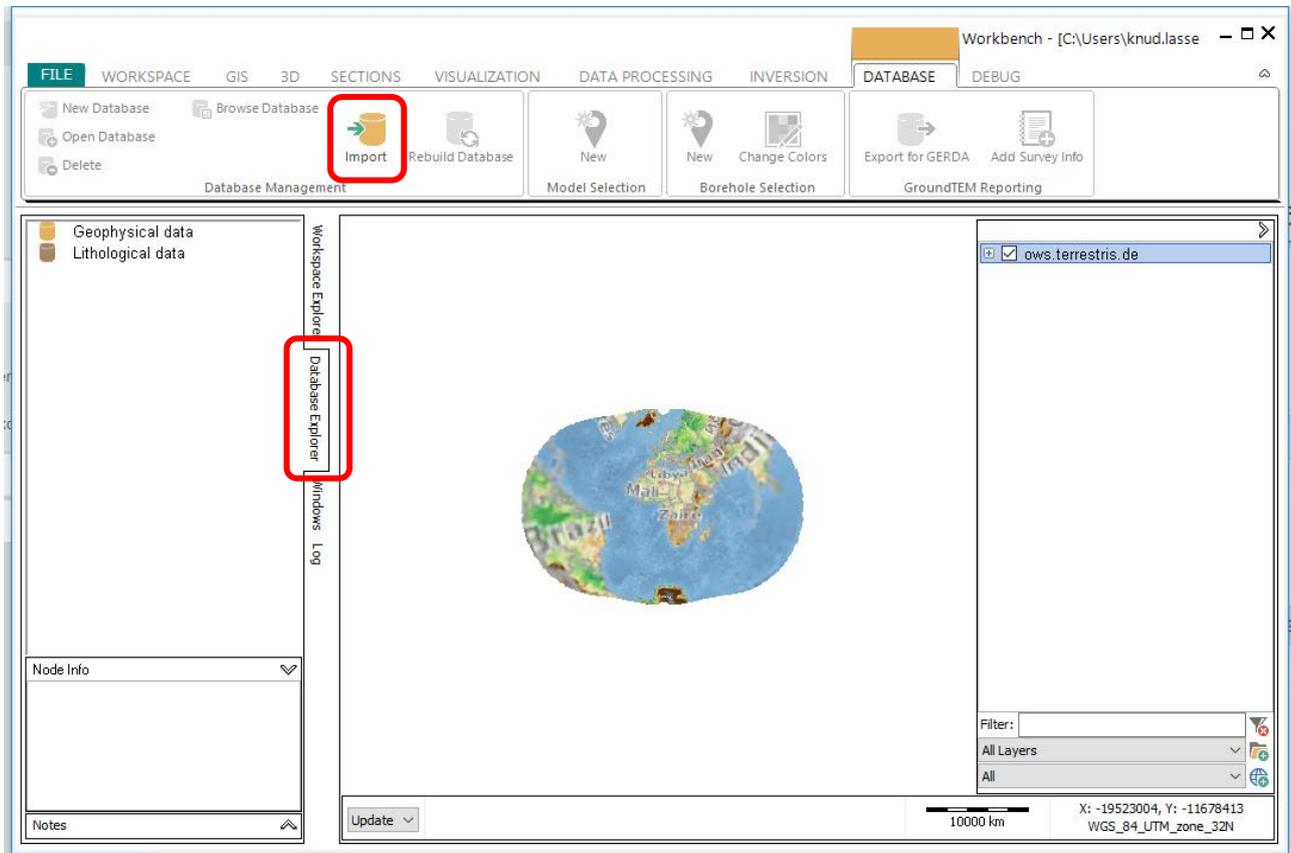


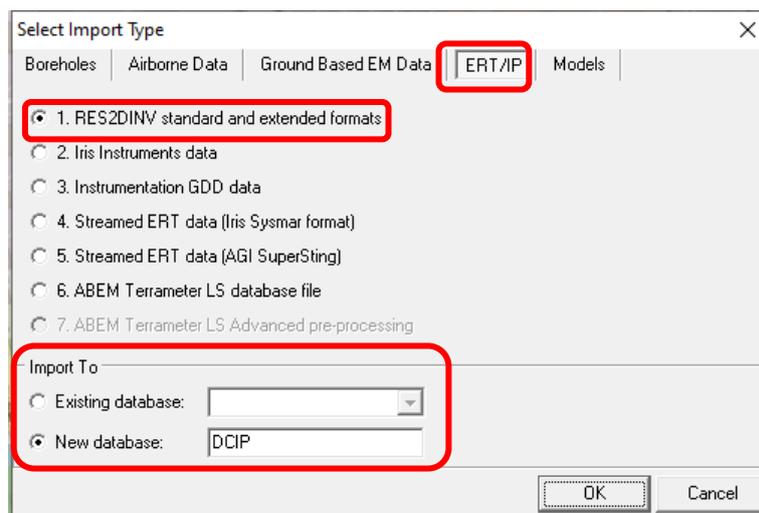
Figure 3 - Initial view of Aarhus Workbench after creating a workspace

Data import

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



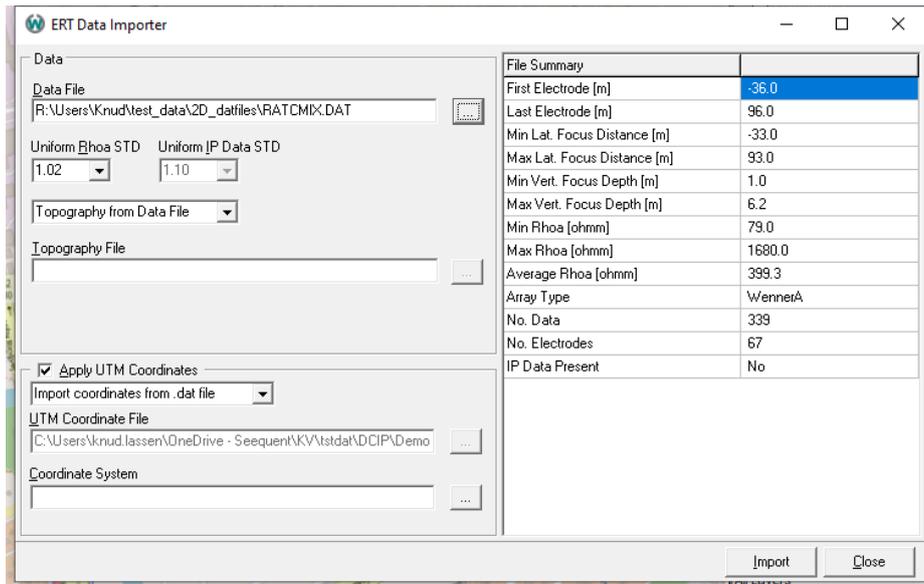
This opens the import window in which “ERT/IP” data is selected followed by “RES2DINV standard and extended formats” (Or one of the other supported formats). If this is the first dataset in the workspace “New database” is selected and a name for the database is entered, if this is not the first import 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. If the data are in the standard .dat format (see the Res2Dinv manual for

specification <https://www.aarhusgeosoftware.dk/res2div-res3div> we recommend using the “general array” version of the format) no other files are needed for the import. Global coordinates and topography can either be taken from the .dat file or specified separately in the .ewp and .ez file, see formats below. If “Apply UTM coordinates” are selected the coordinate system that is used for specifying locations in the .dat file must be entered as well. If the checkbox is unchecked dummy coordinates will be used.

After selecting the appropriate files press “import”, select the name for the data line and to which map node it is added when prompted. Your data file is now imported in the workspace and you can continue by importing additional files or go on to processing and inversion.



Format of the .ez topography file

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.

Format of the .ewp coordinate file

The .ewp file contains coordinates for the ERT profile made and must be made manually. The file needs to contain at least two coordinates. It does not have to be coordinates for the first and last electrode in the profile, it can be for any two electrodes. The importer in Aarhus Workbench will automatically make a linear interpolation for the remaining electrodes in the profile. The more coordinates that are described in the .ewp file the more precise the interpolation will be.

If the profile is not a straight line but makes a turn, this can be accounted for by setting Angcon to 1 at the electrode where the turn starts. An example of a .ewp file is listed below including comments for each line.

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. Interpolates between z coordinates for remaining electrodes.
1	No	utm _x	utm _y	std _x	std _y	edist	Stde	Angcon	utm _z
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

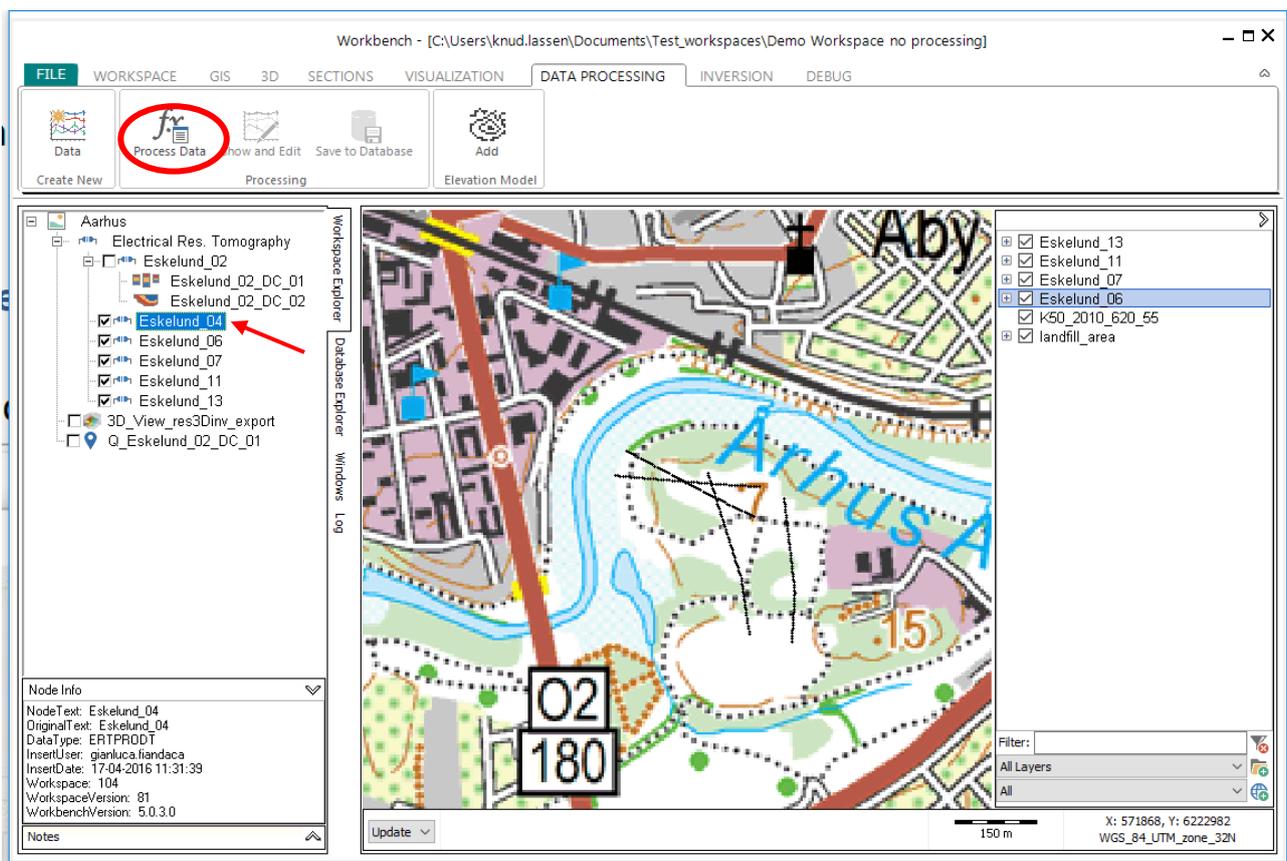
Guide for processing and inversion of ERT data

The workflow presented in the following will give good results for most ERT 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 ERT data can be found in the video tutorials here: http://www.ags-cloud.dk/Wiki/W_ERTVideoTutorials.

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 main tools in the DC data processing are the pseudo section and the data profile (Figure 4).

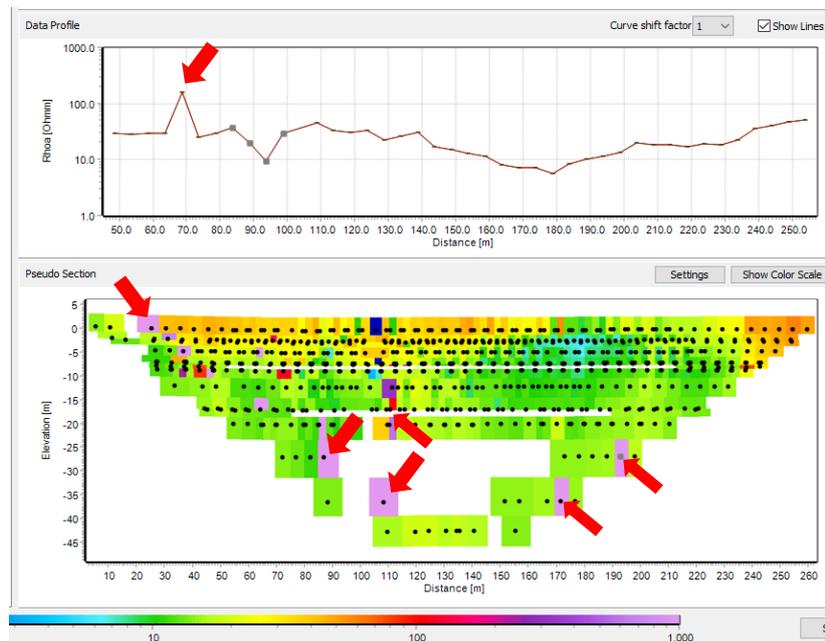


Figure 4

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 electrodes, instrument malfunction etc.

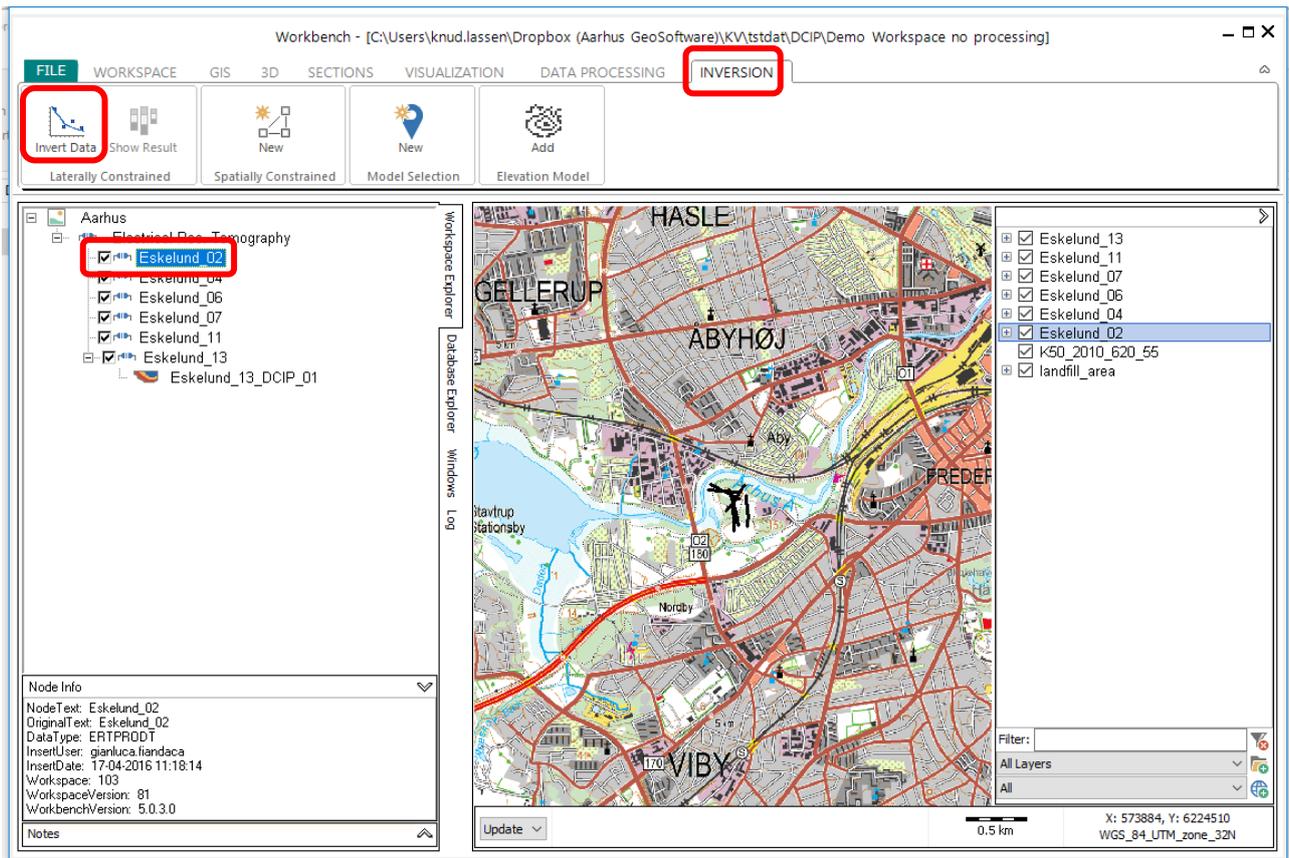
Data points are toggled in and out of use by pressing **Alt+A** and **Alt+Q** while the points are selected with the mouse. It is also possible to remove all datapoints in which a bad electrode is involved (e.g. due to bad electrode contact or faulty cables) by selecting the electrode in the “Electrode Positions” window.

DC inversion and reprocessing

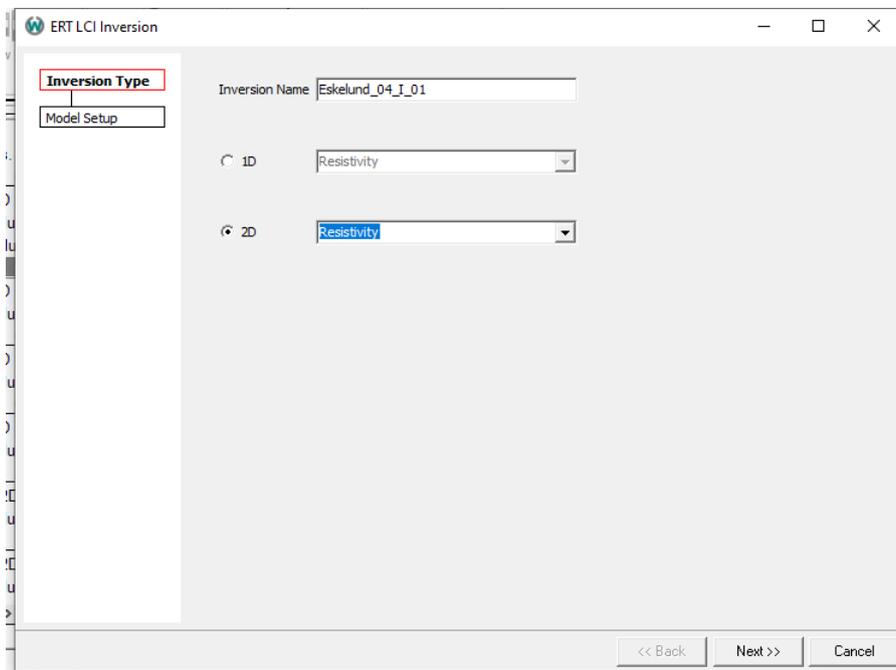
The next step is to run a DC inversion and evaluate the data fit using the DC residual pseudo section. This is primarily a task of either removing or assigning a higher standard deviation to data points with a high misfit, and to evaluate whether the chosen constraints and model discretization is suitable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

Running a DC inversion

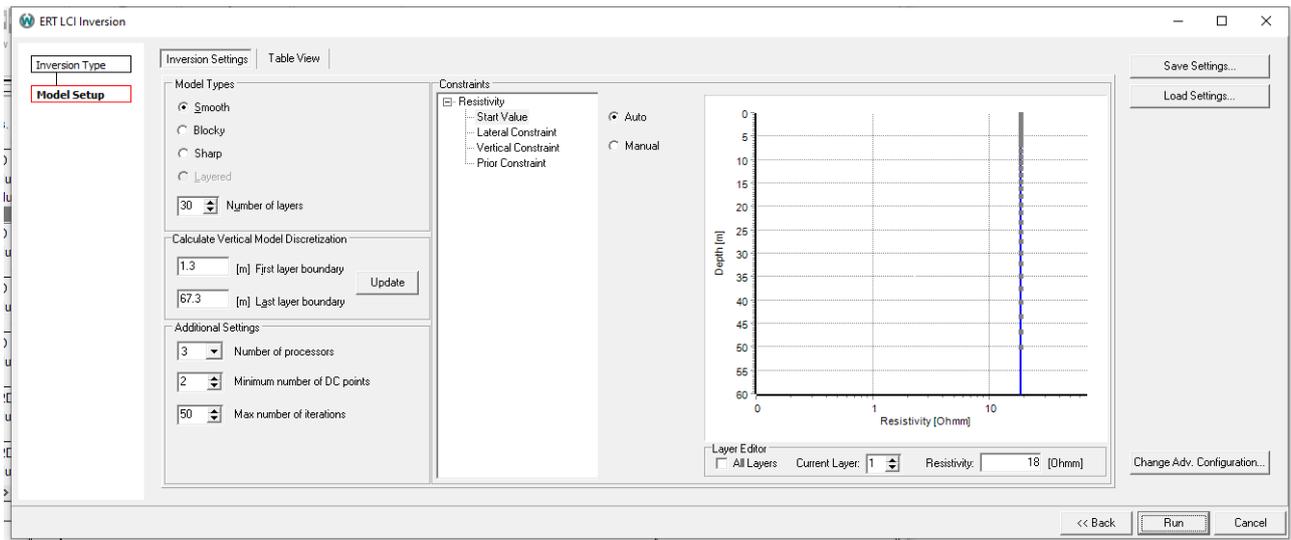
A DC inversion is run by highlighting the inversion node and pressing “Invert Data” in the “INVERSION” ribbon:



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

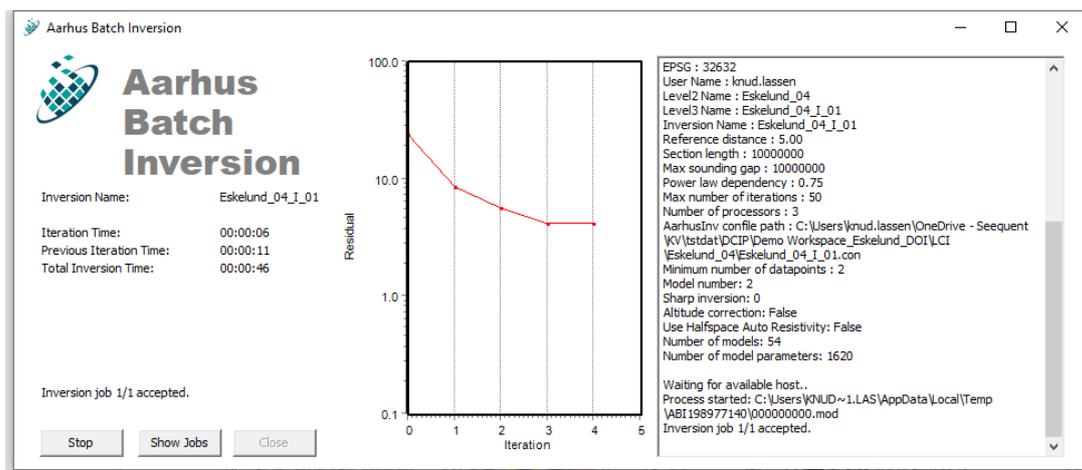


In this case a 2D resistivity inversion is used as the basis for further processing. Selecting “Next” will open a new window asking for the name of the inversion after which a second window will open in which a number of inversion settings can be specified:



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.

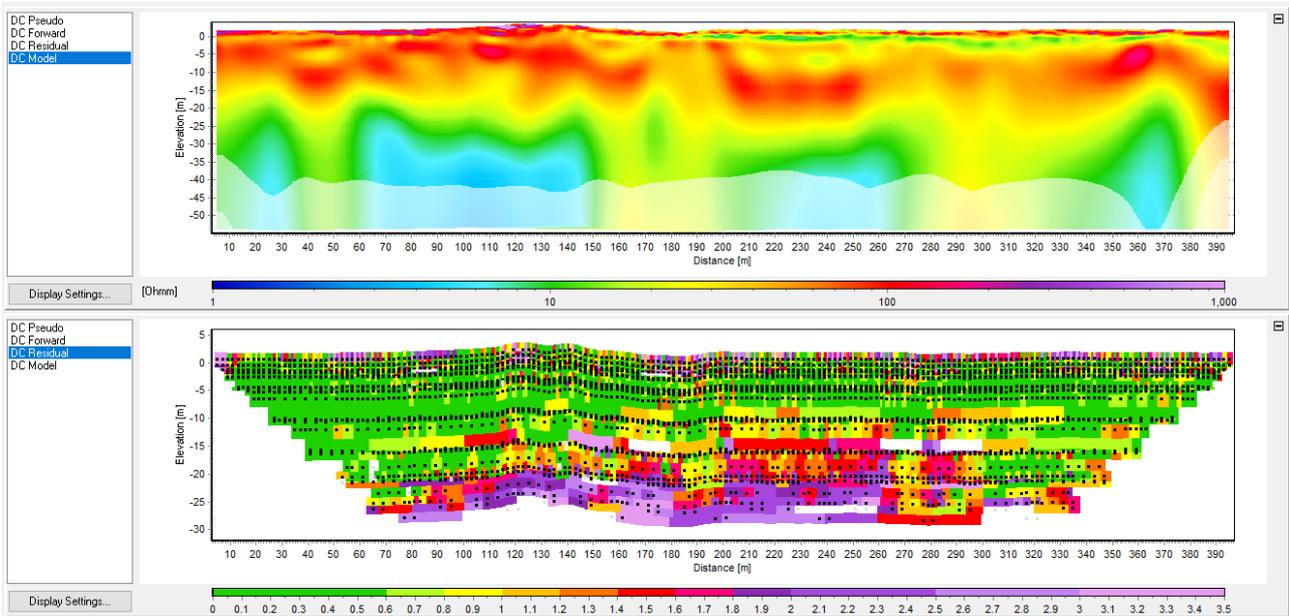


Figure 5 - total residual = 1.11, data example kindly provided by VIA University College, Horsens, Denmark.

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.

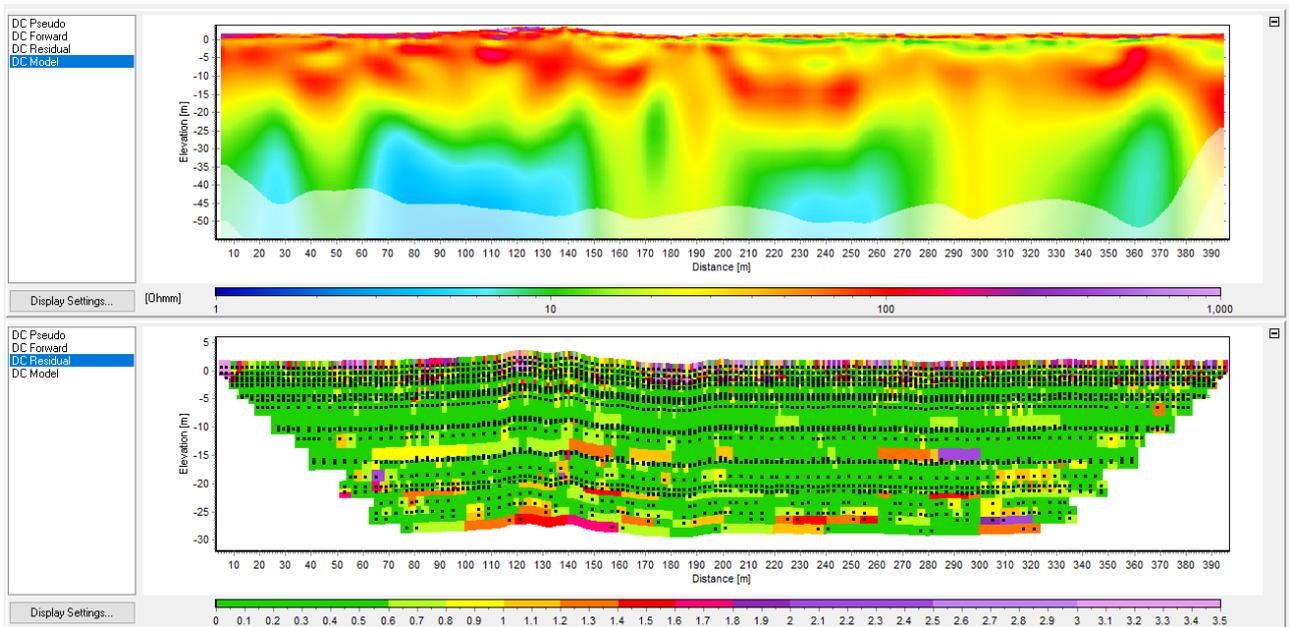


Figure 6 - total residual = 0.86, data example kindly provided by VIA University College, Horsens, Denmark.

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.

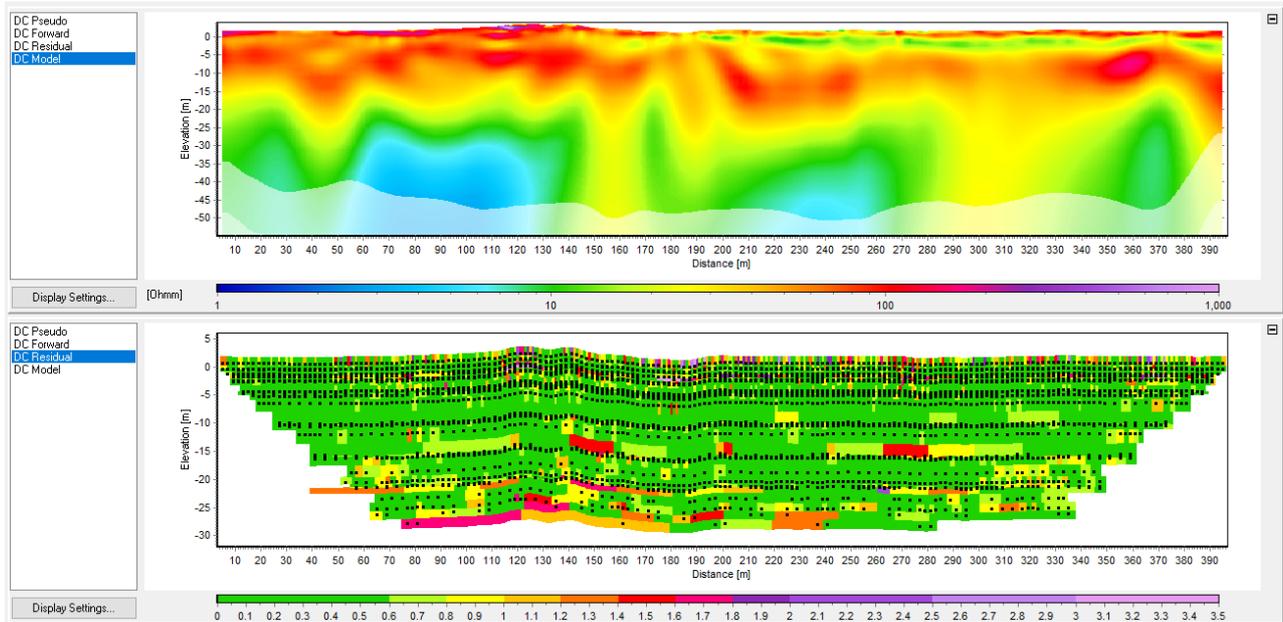


Figure 7 - total residual = 0.59, data example kindly provided by VIA University College, Horsens, Denmark.

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 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 8) 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.

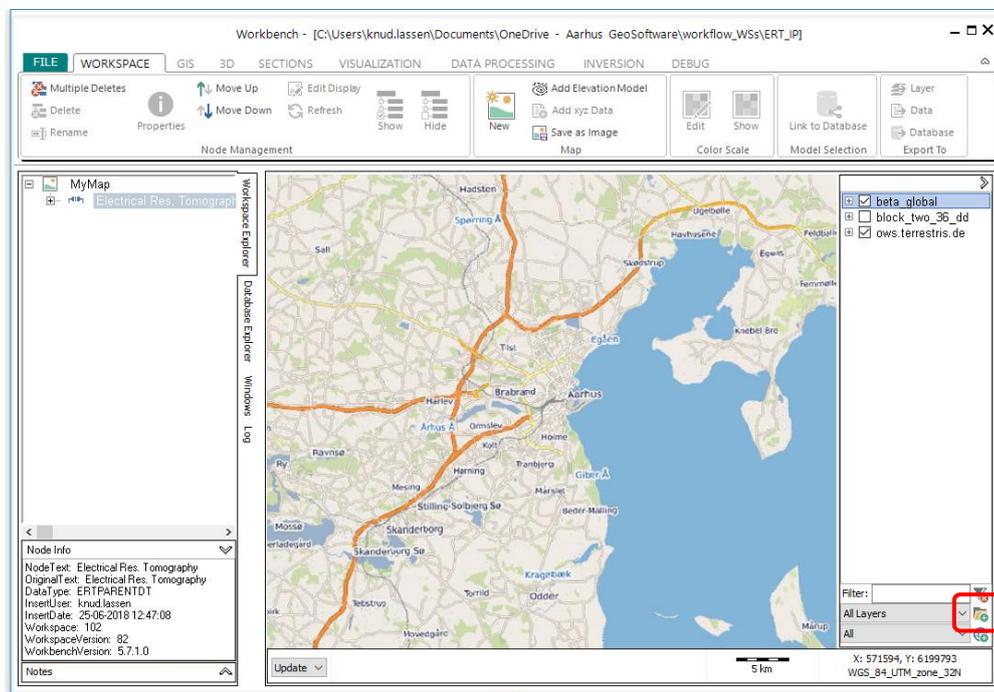


Figure 8

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 8, 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.

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.

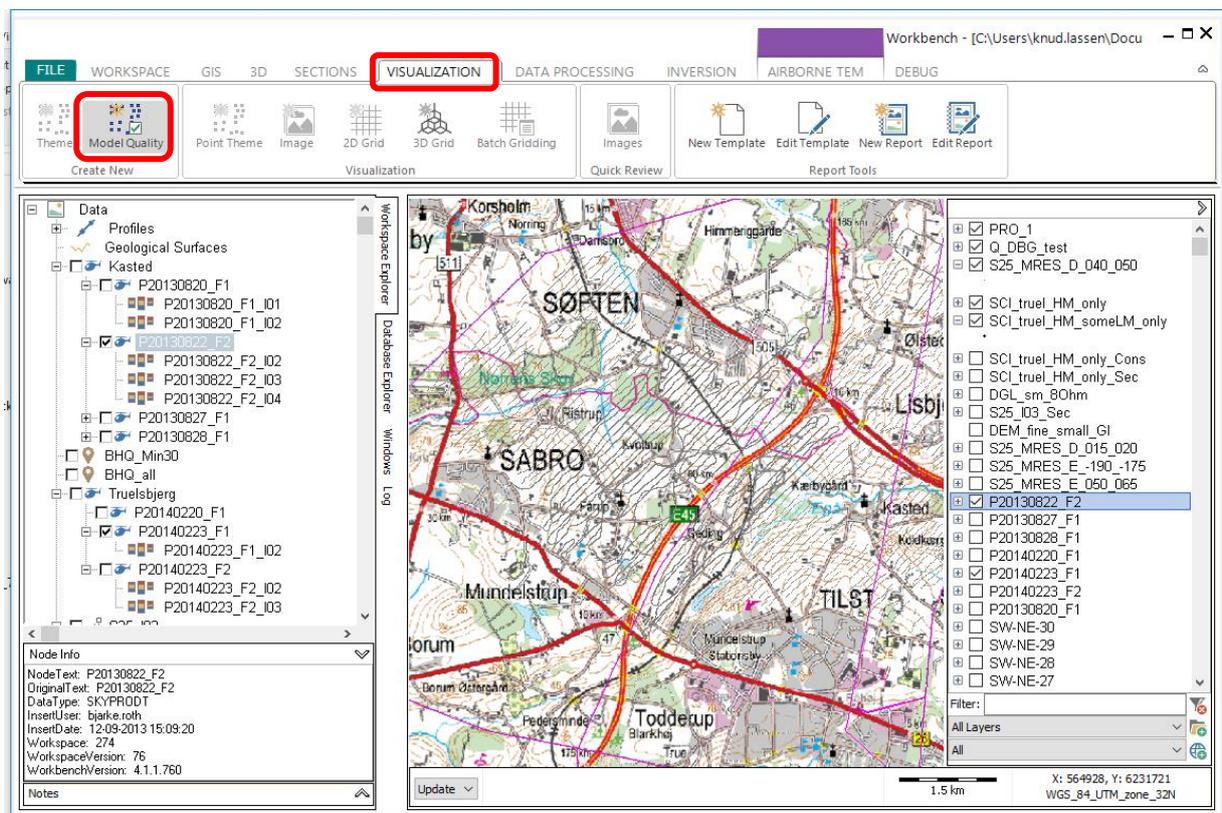
Common values to plot for the QC is total residual, data residual and dept of investigation (DOI).

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.

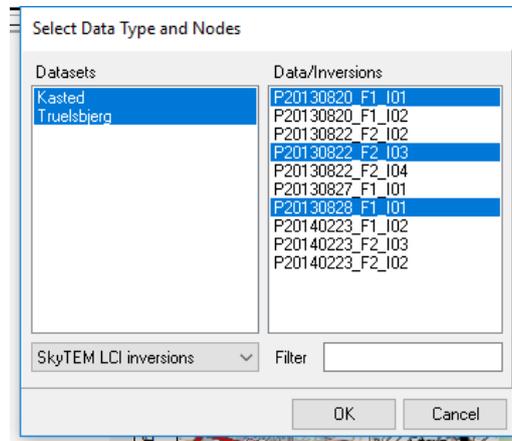
DOI can be used to determine whether the survey resolve the subsurface to a sufficient depth to serve the surveys purpose, and to guide the interpretation of data.

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.

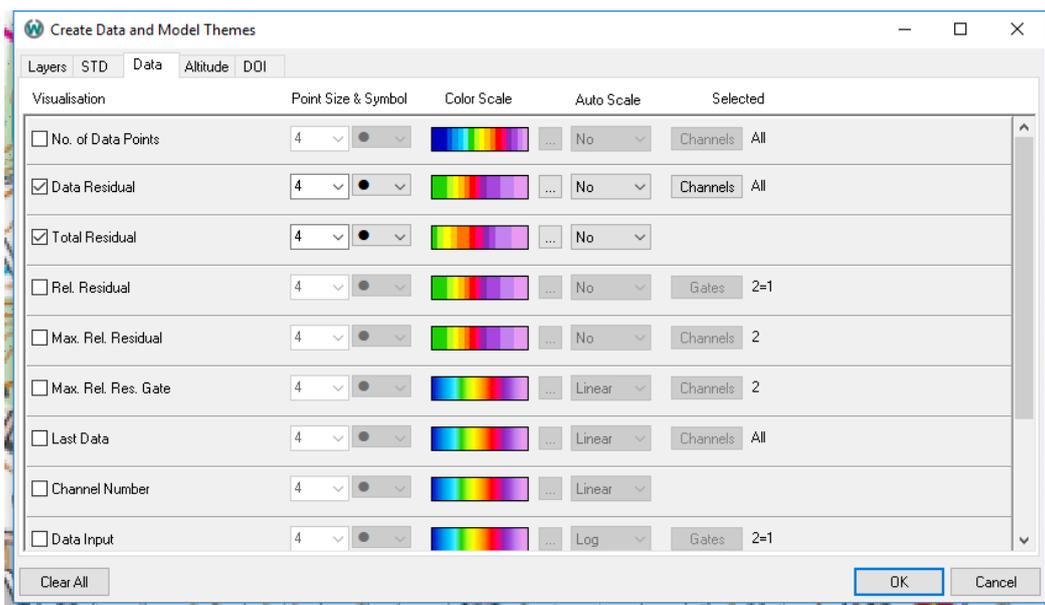
Model quality themes are created by navigating to the "Visualization" ribbon and selecting the "Model Quality" button:



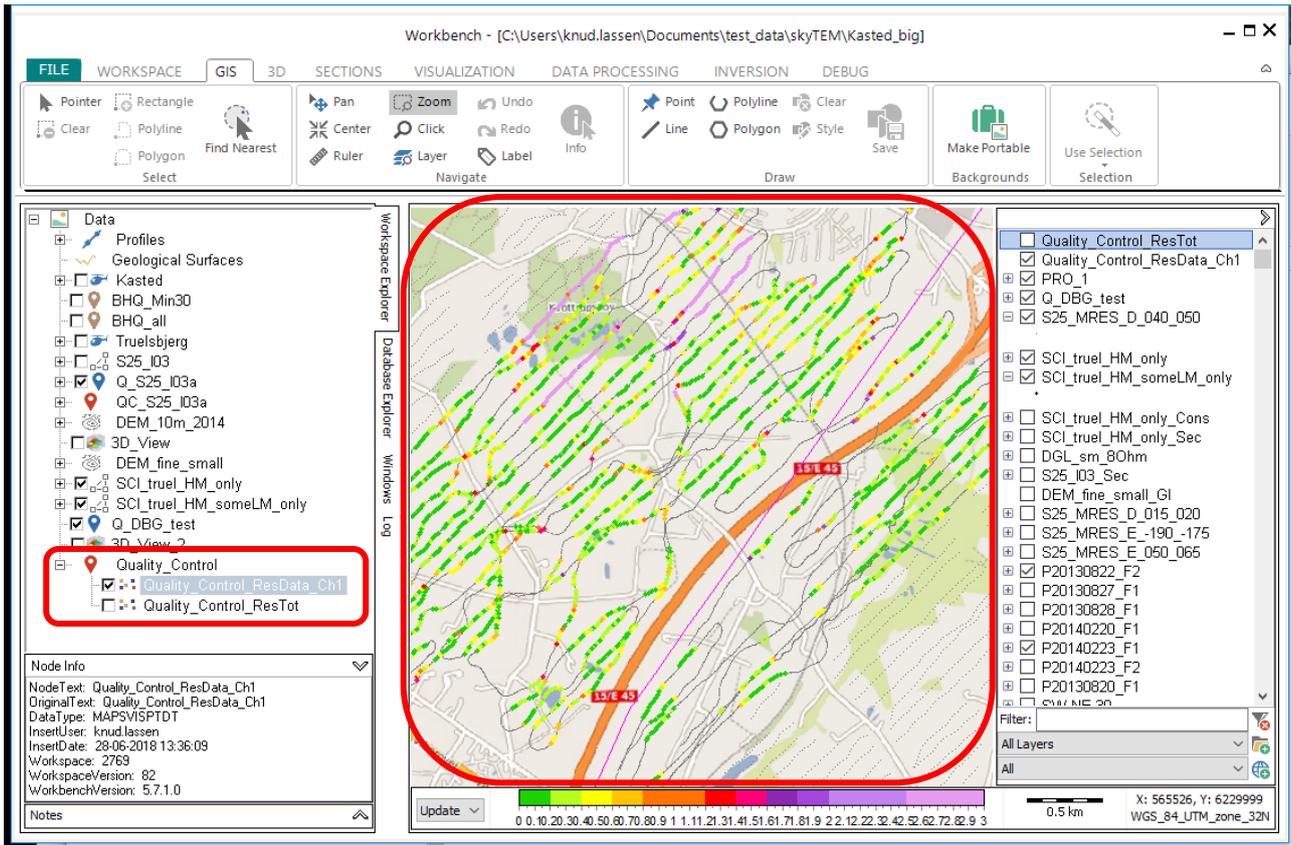
In the resulting dialogue box, it is possible to select which inversions from which datasets to work with. Note that more than one dataset and inversion can be selected by holding down “CTRL” while selecting:



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:

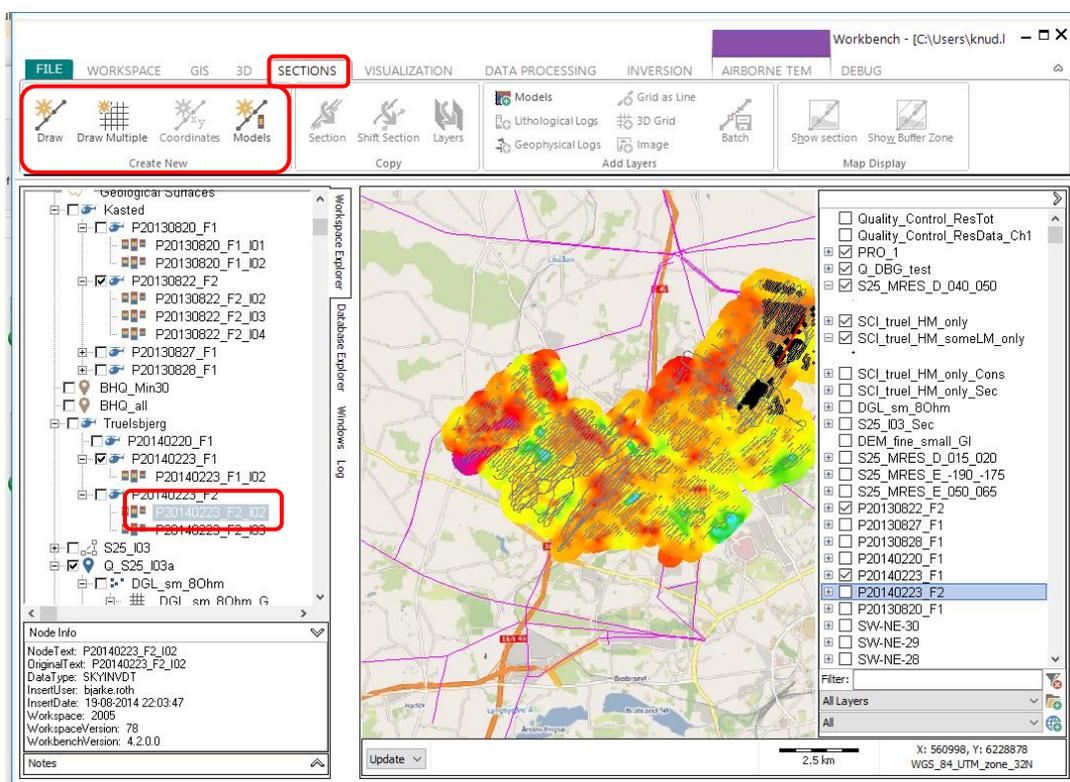


Create sections and add models and data to these

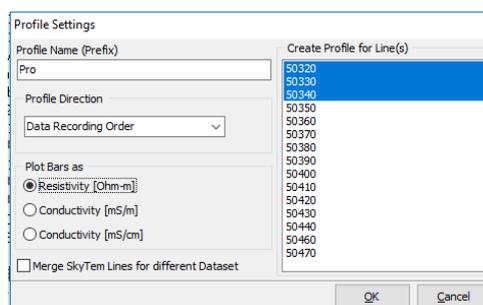
Sections are the classical way to display inversion results from different inversions together with borehole data, logs, geological surfaces etc.

Sections can be created in two ways:

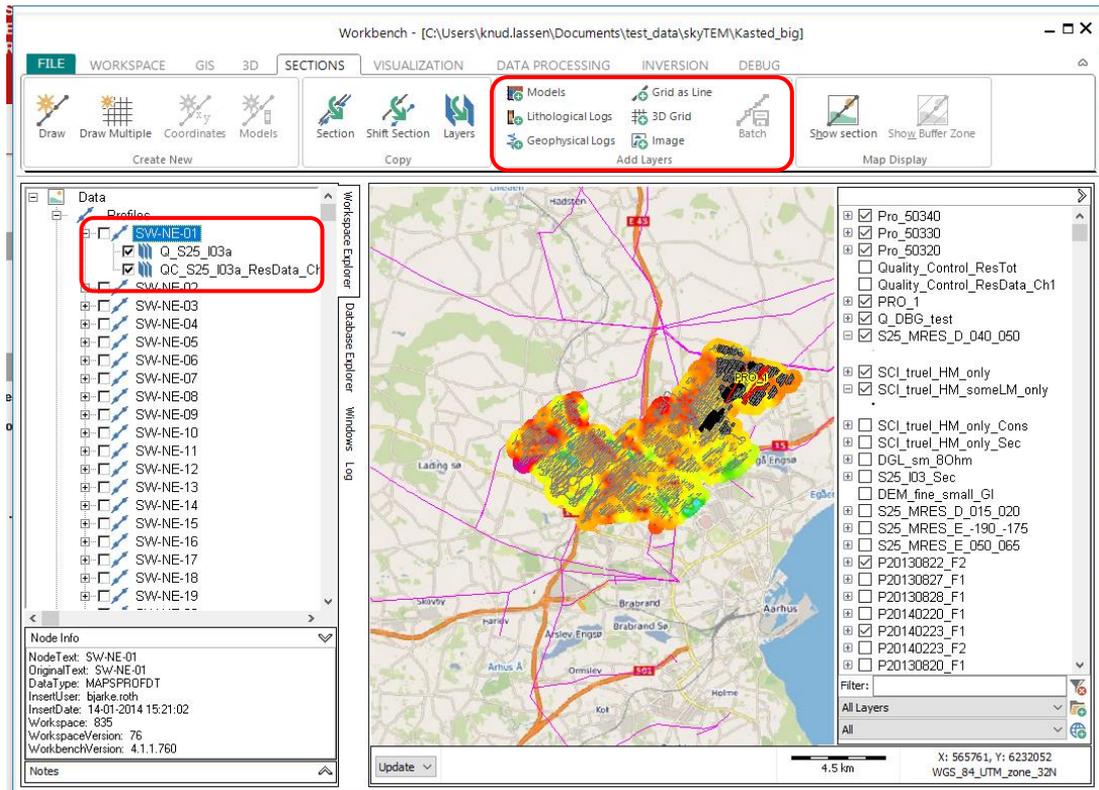
1. Under the “Sections” ribbon in the “Create new” group “Draw” can be selected, and the section can then be drawn in the GIS, single click for each bend in the section and double click to end the drawing and. After double clicking you will be prompted to name the section.
2. Draw section from inversion.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



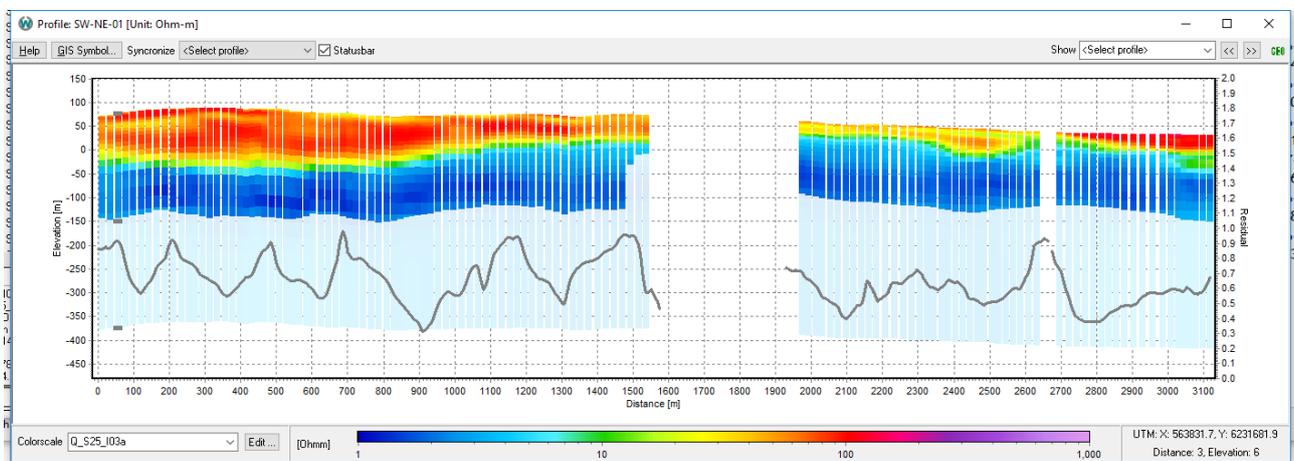
If the inversion node contains more than one data line you will be prompted to select for which lines you want to create sections, you can also select the name and how to plot the data at the section:



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:



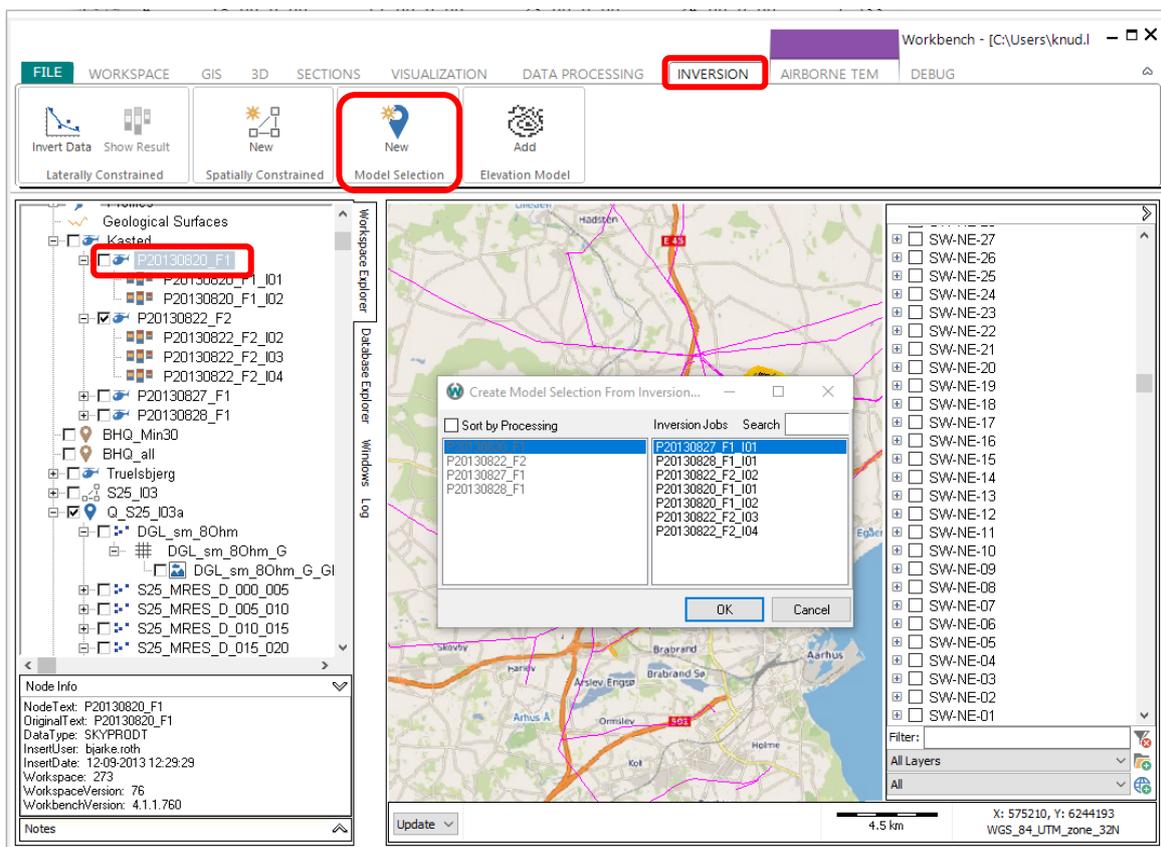
Note that the range and type (logarithmic or linear) of the axis can be edited by double clicking the axis.

Creating and using themes

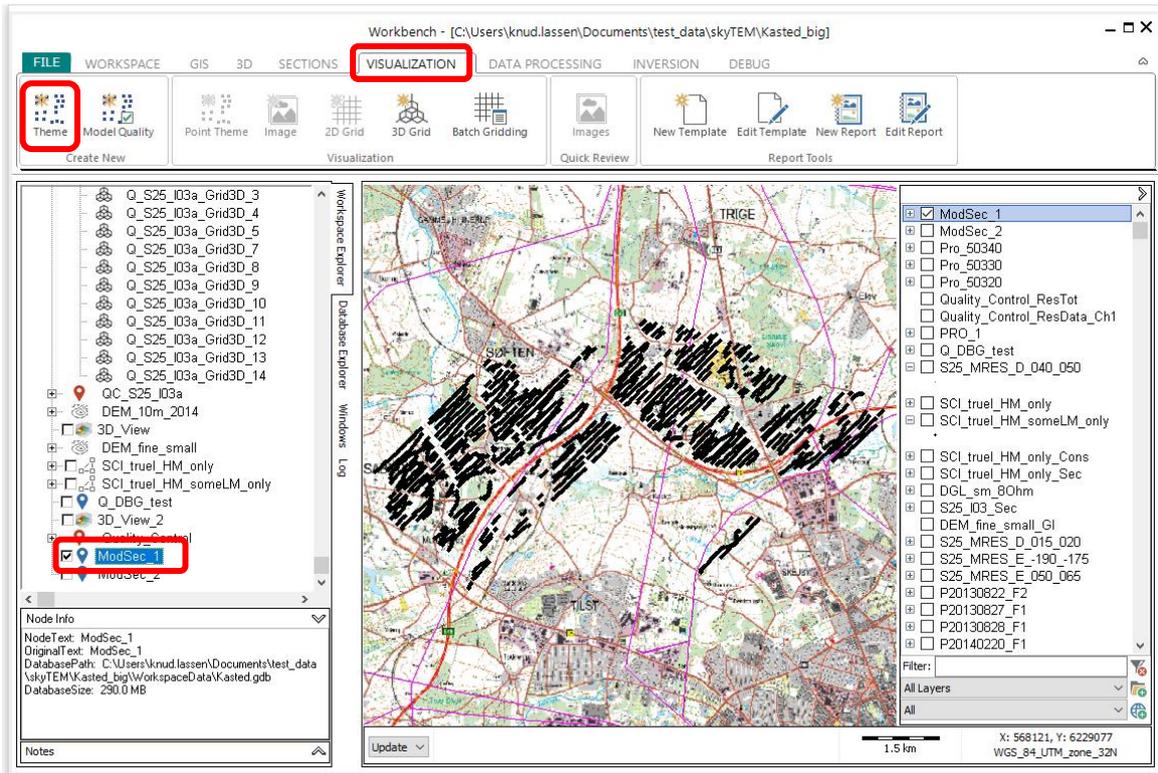
Themes are the tool for visualizing results on the map e.g. the mean resistivity of a given depth interval, the depth to a good conductor, the resistivity of the third layer in the model, or thickness of a body with a given resistivity.

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).

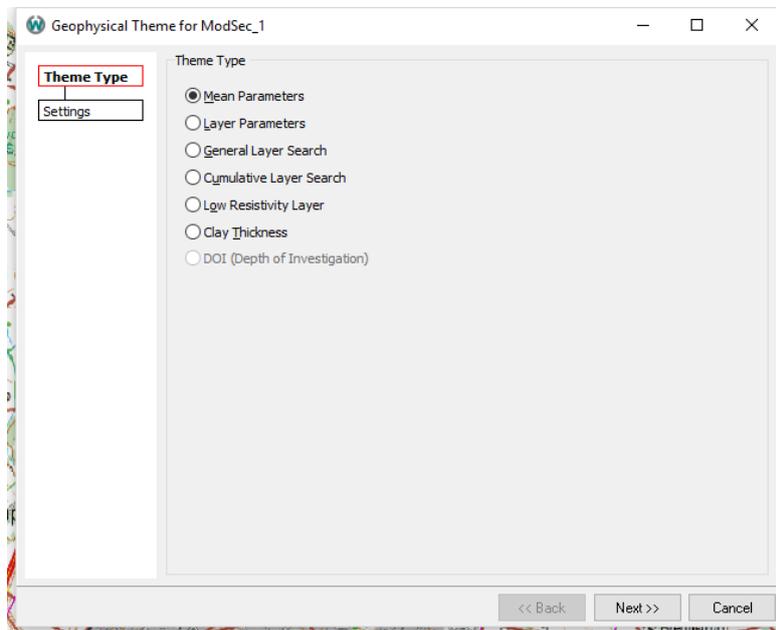
A theme is created from a model selection. A model selection is created from the “Inversion” ribbon by highlighting a processing or inversion node and selecting “New” from the “Model Selection” group. A model selection can contain models from inversions from different processing nodes by removing the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



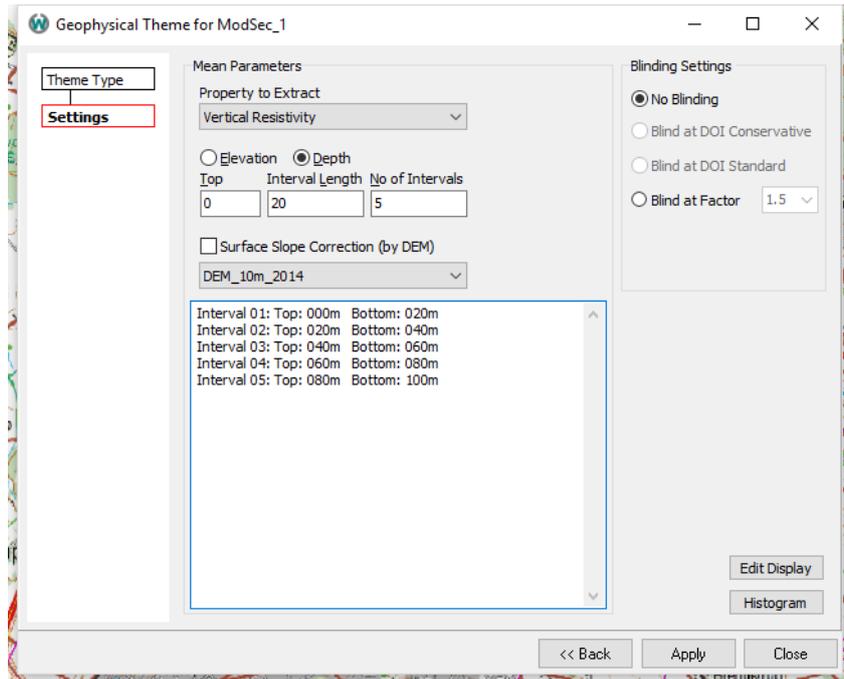
After creating the model selection, the actual theme can be created by highlighting the model selection in the workspace explorer and selecting “Theme” in the “Visualization” ribbon:



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:

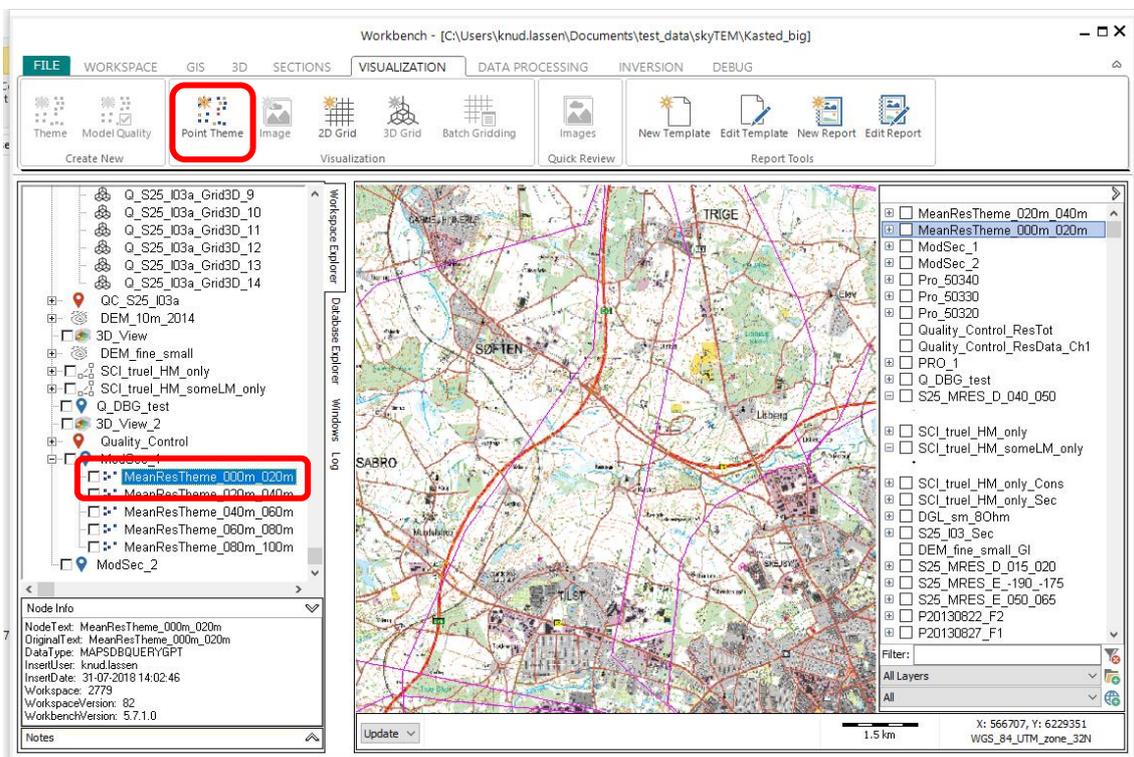


The themes are created by selecting “Apply”. The themes can now be visualized as points or 2D grids.

Visualizing themes

Points

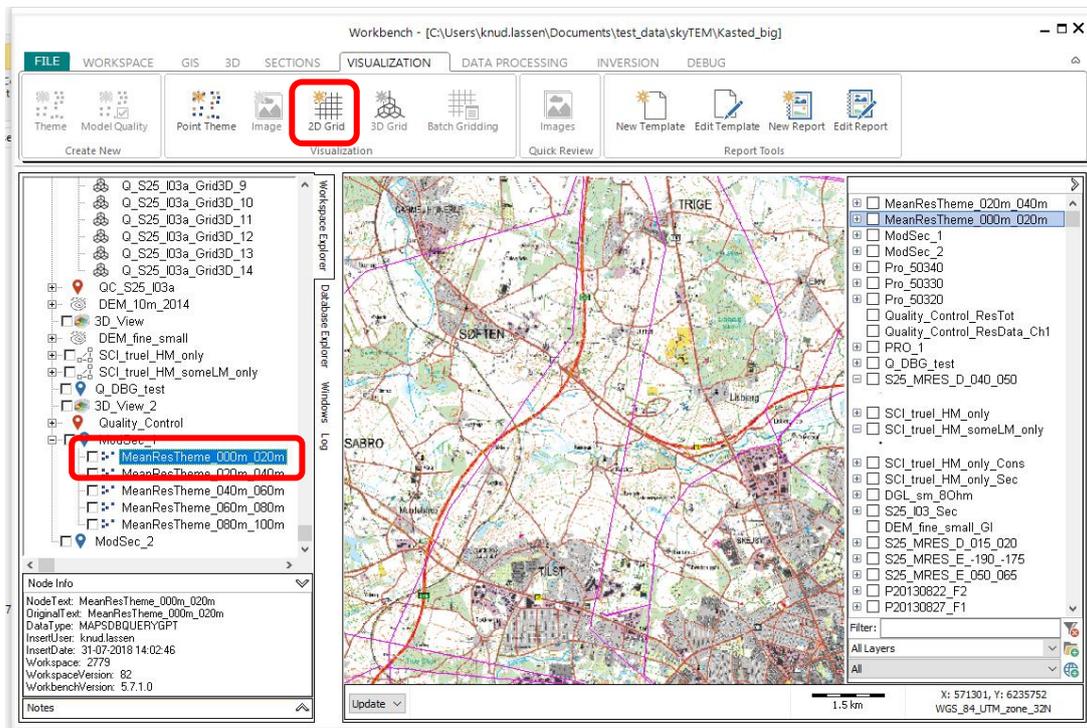
To plot the gridded values as colored points in the GIS window highlight the theme and press “Point Theme”



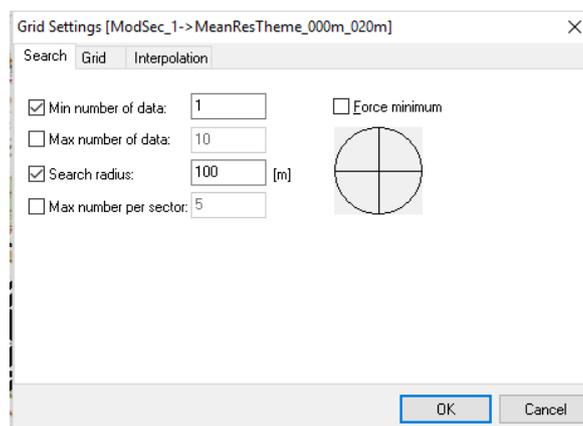
Select the color scale, point size and point shape in the dialog box and press “OK” and select the name of the theme, the point theme can now be found in the workspace explorer.

2D Grids

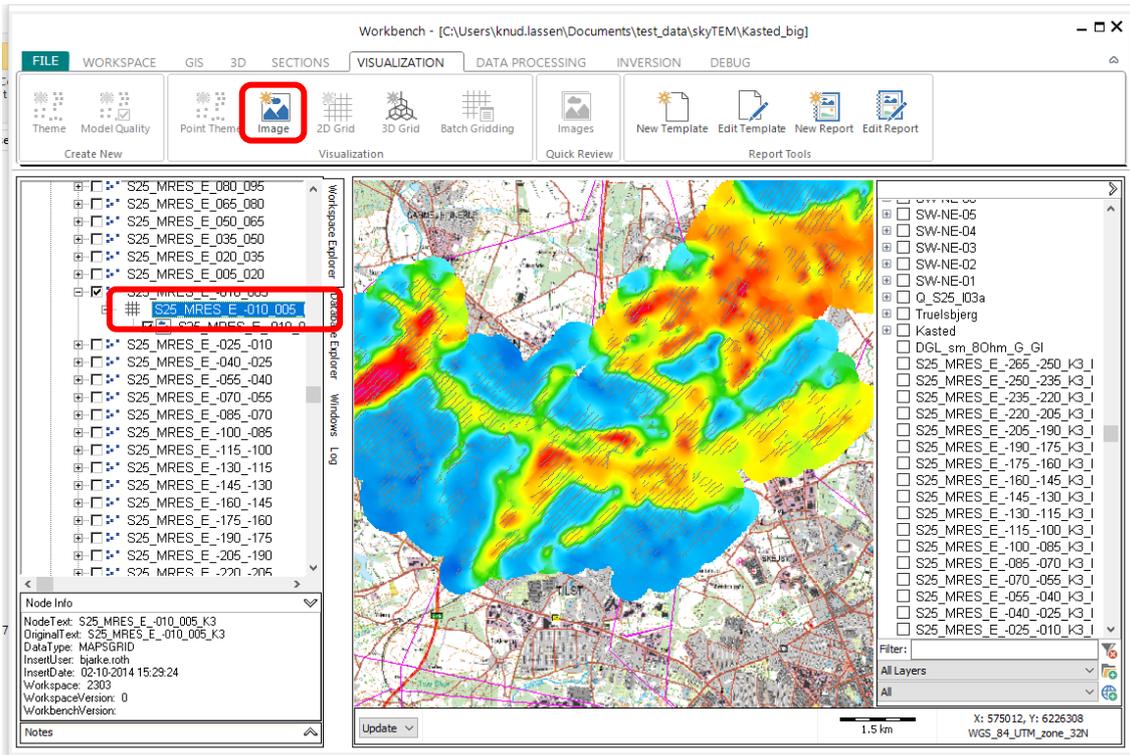
The other way of presenting themes are as surface covering interpolated grids, these are created by highlighting the theme and pressing “2D Grid”



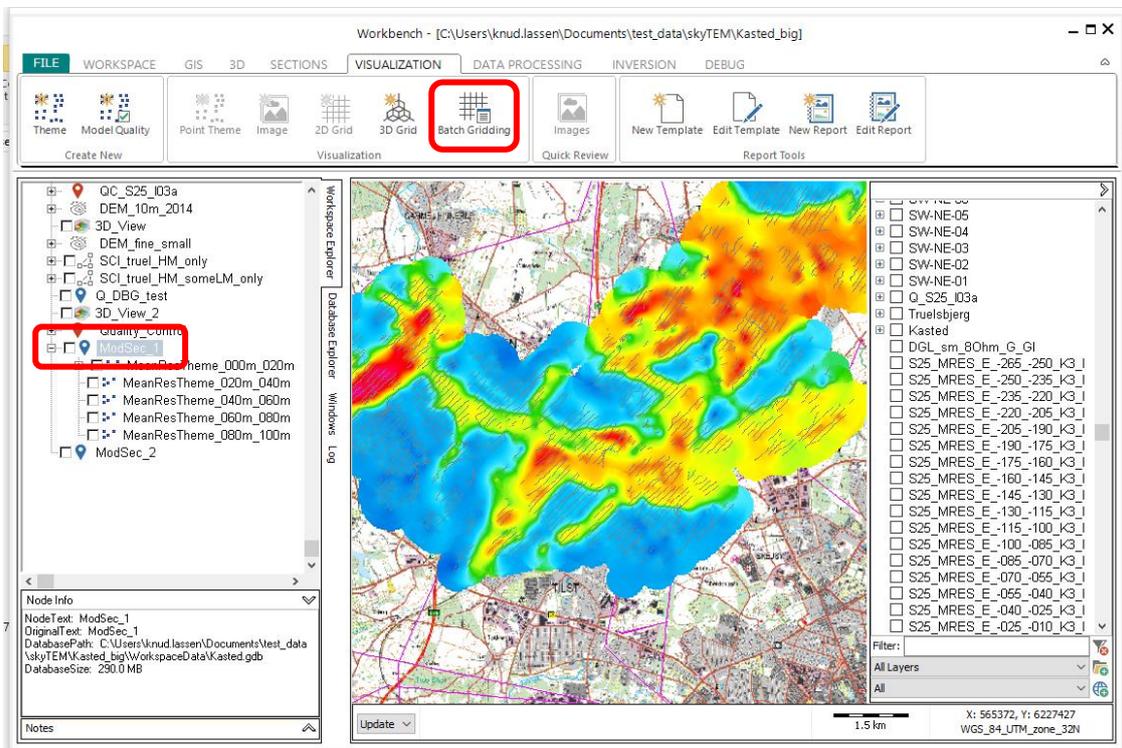
The following menu box will appear:



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”:

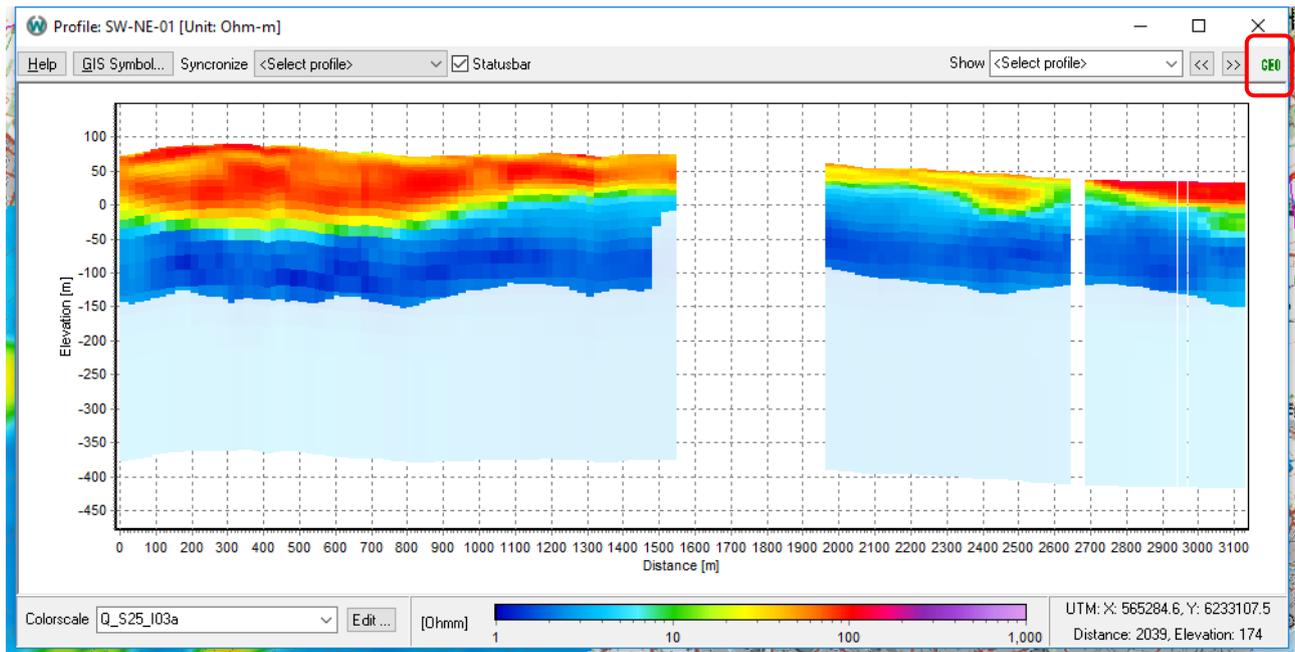


Drawing and gridding geological surfaces

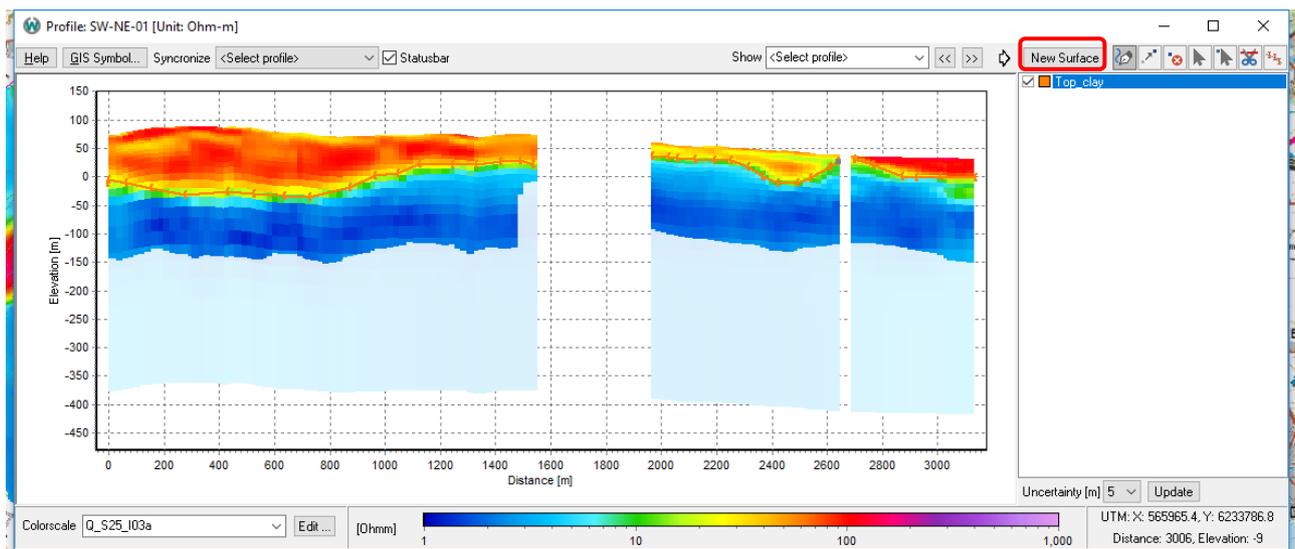
Geological surfaces are the interpretation tool of Aarhus Workbench, these can be used for tracking geological bodies and other targets, such as the interface between sand and clay, the top and bottom of a buried landfill or the water table in an aquifer, across different sections and data types.

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” button 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.

Uncertainty [m] 5

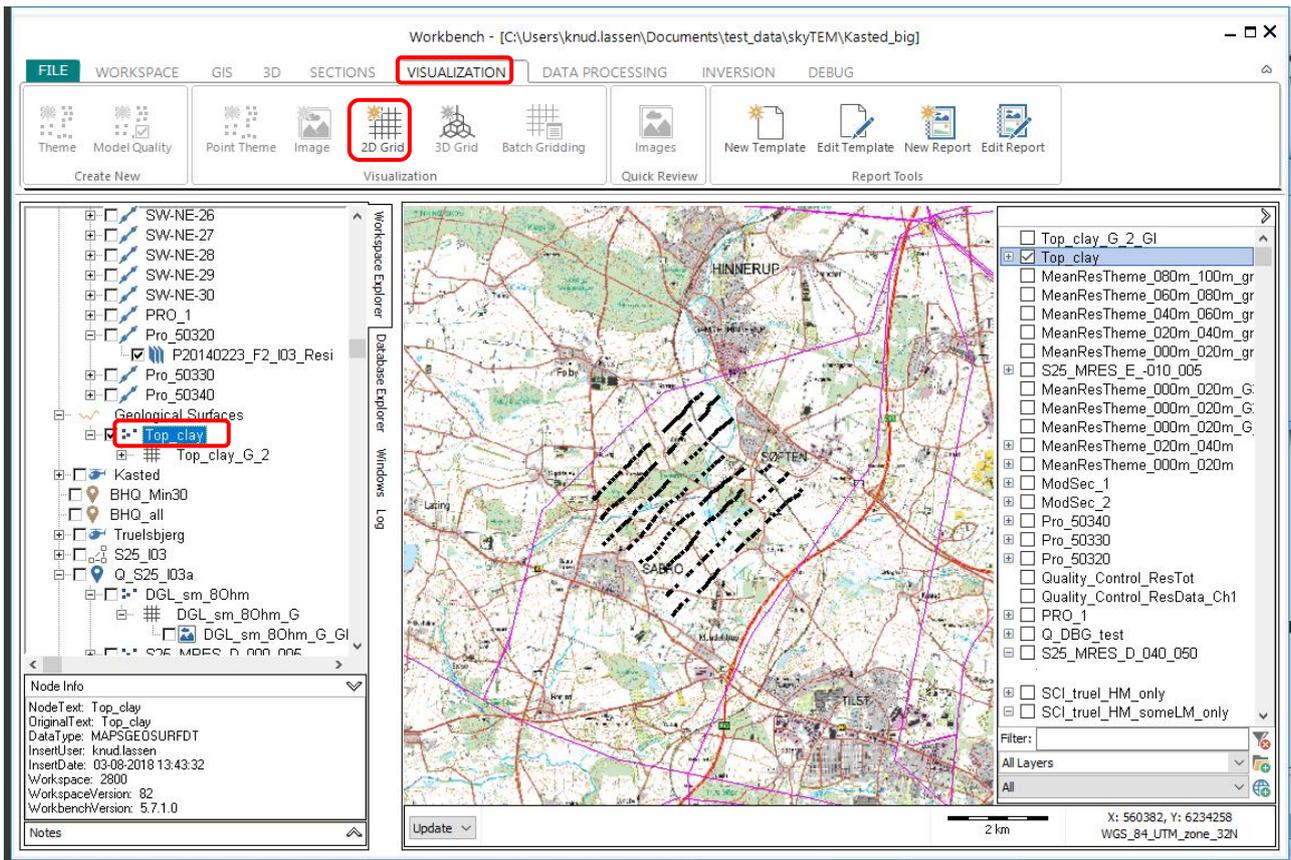
Change uncertainty, changes the uncertainty on the selected points when pressing “update” and on any new points drawn, this feature makes it possible to express the uncertainty on the exact location of the geological surface.

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”.

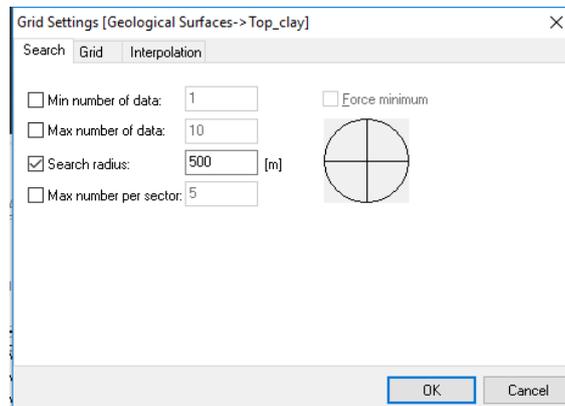
The location of the points on the geological surface can be shown on the GIS map by checking the name of the geological surface in the workspace explorer under “geological surfaces”. This can be useful in getting an overview over which surfaces are drawn for which profiles. Before this is possible it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

Gridding of geological surfaces

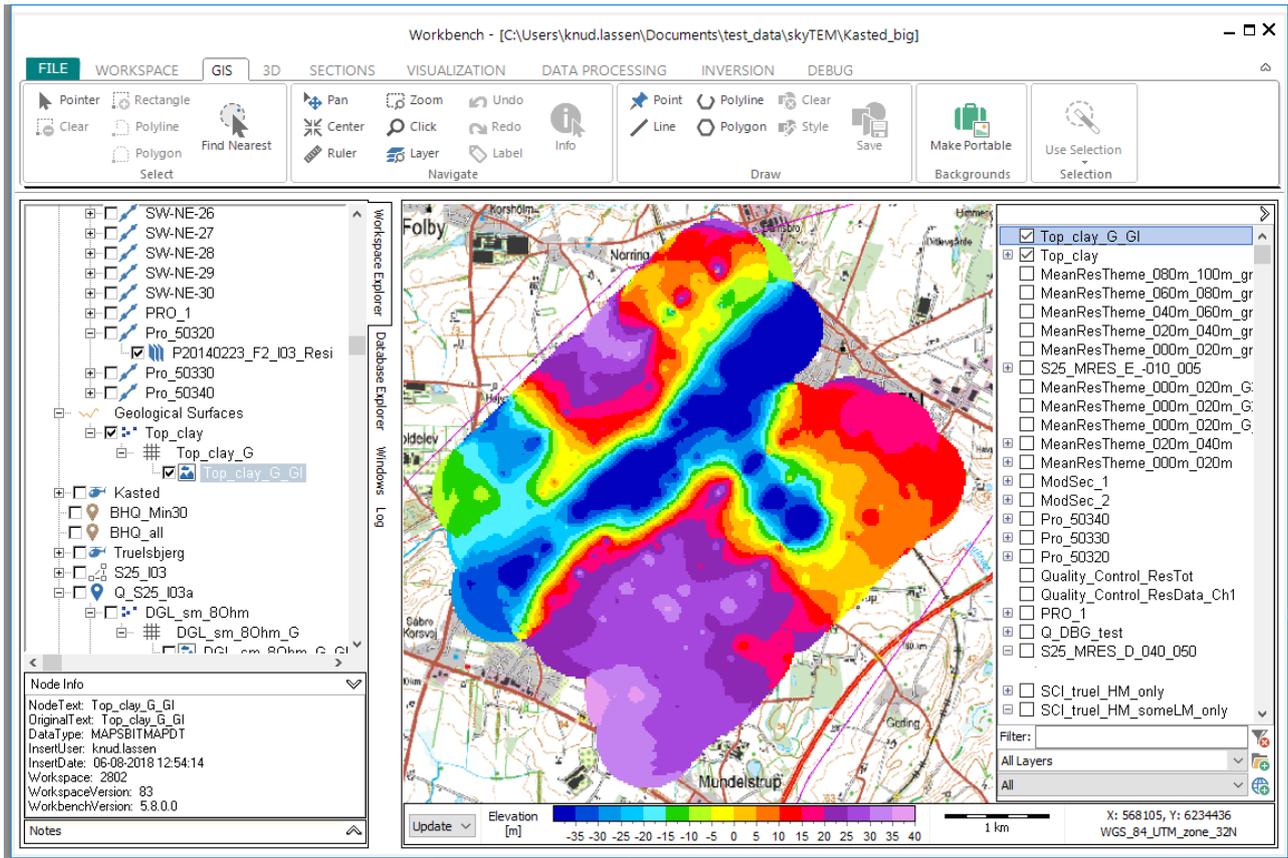
Once the geological surface is drawn on all sections/profiles in the area of interest the surface can be gridded to be shown on a horizontal map. The grid is created by navigating to the “VISUALIZATION” ribbon, highlighting the geological surface to grid in the workspace explorer and pressing “2D Grid”.



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.



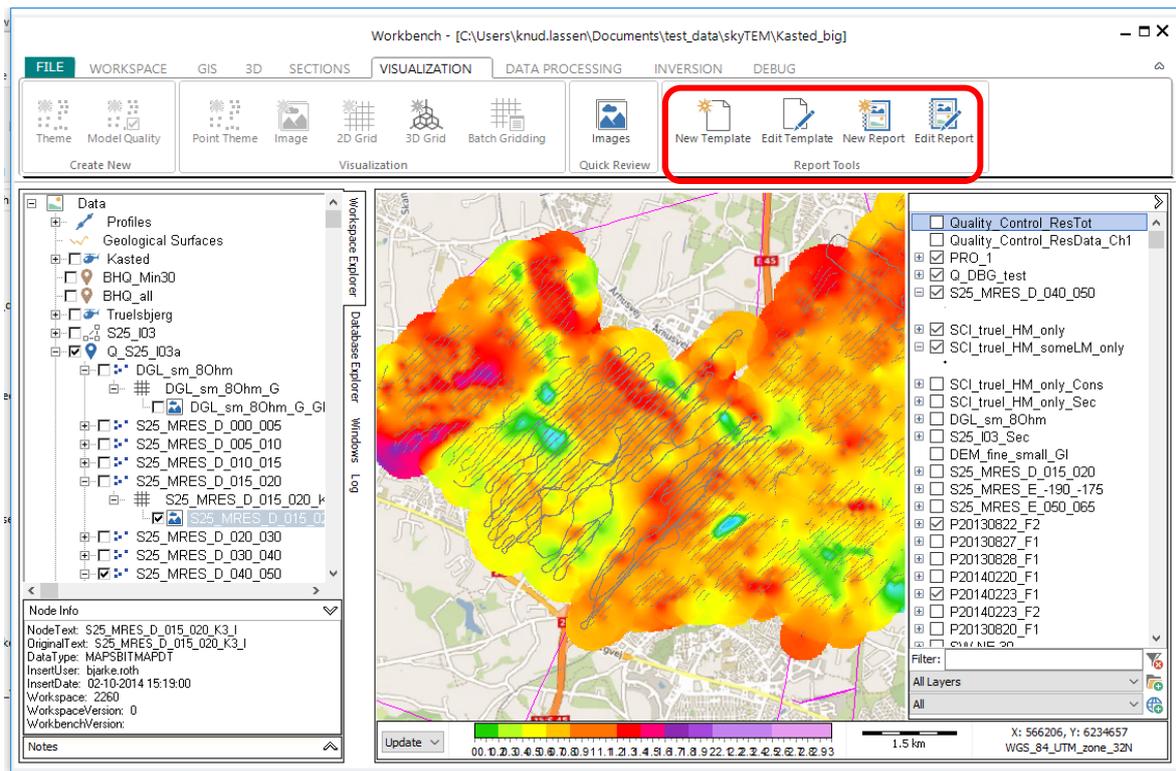
Once the grid is created it can be found in the workspace explorer under the geological surface, to display the grid on a map an image must be created for the grid. This is done by highlighting the grid in the workspace explorer and selecting “Image” under in the “VISUALIZATION” ribbon, you will then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



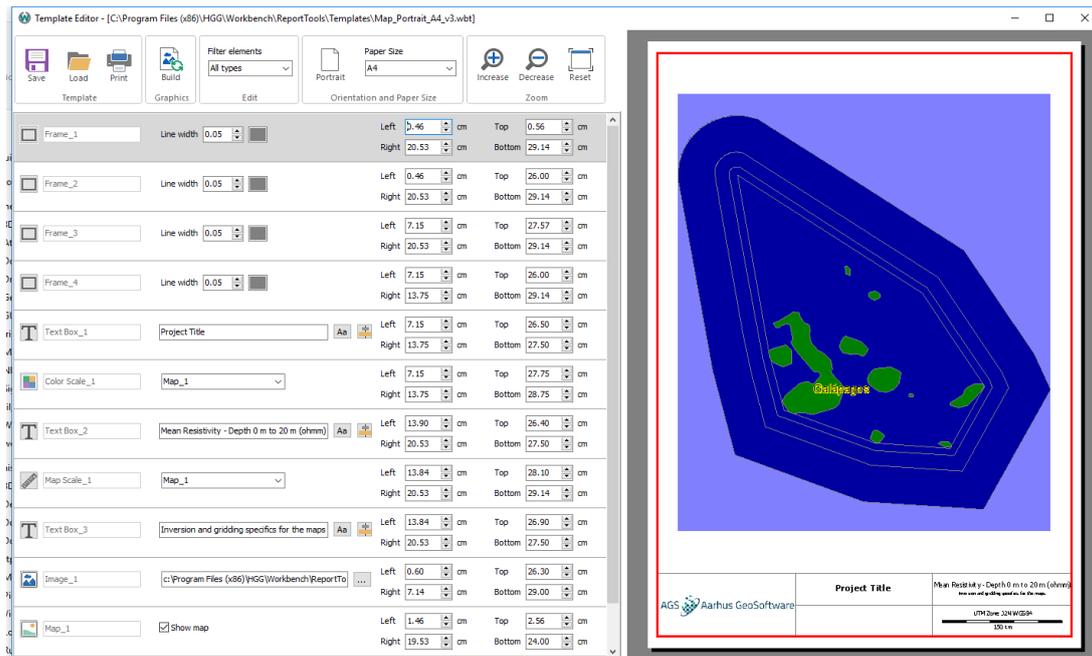
Creating a PDF report from template

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:

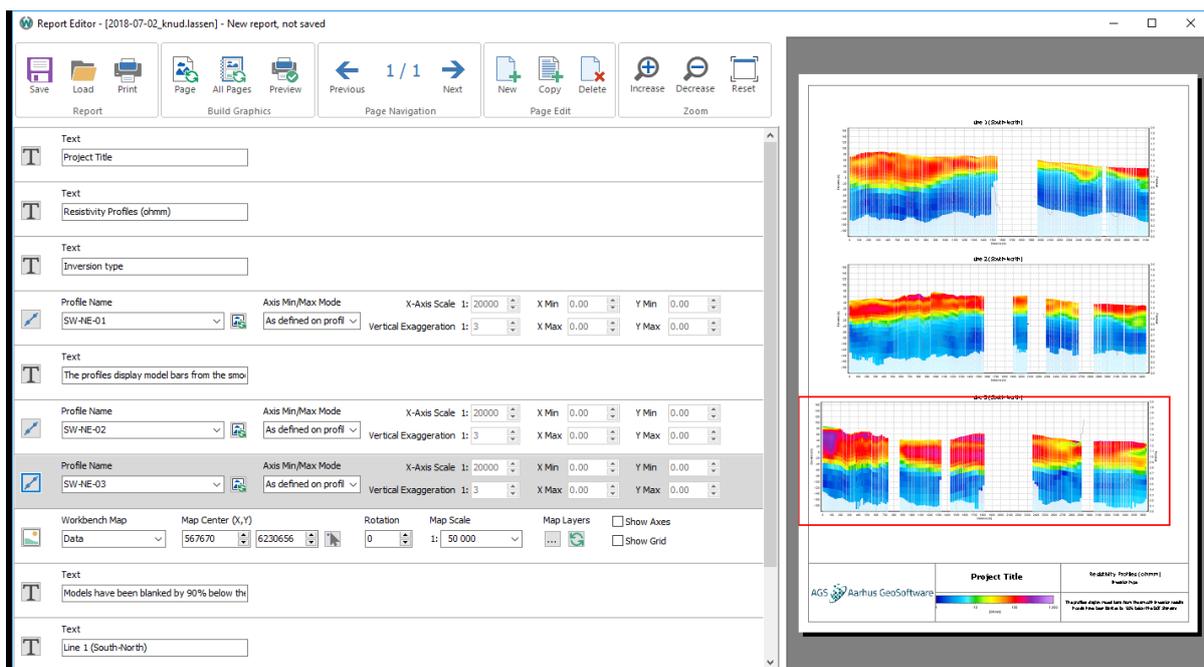


Each report page is built from the following elements: frames, text boxes, maps, legends, map scales, north arrows, images, profiles and color scales. By selecting either new template or edit template the template editor is opened:



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.



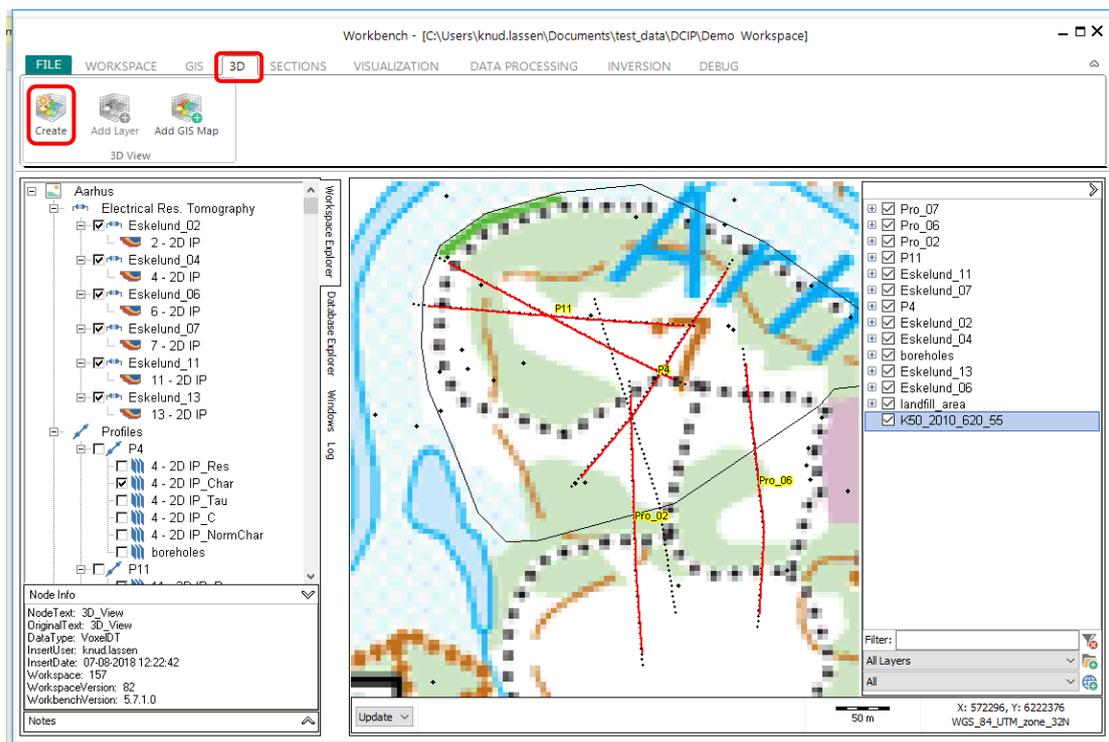
Using the 3D viewer

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

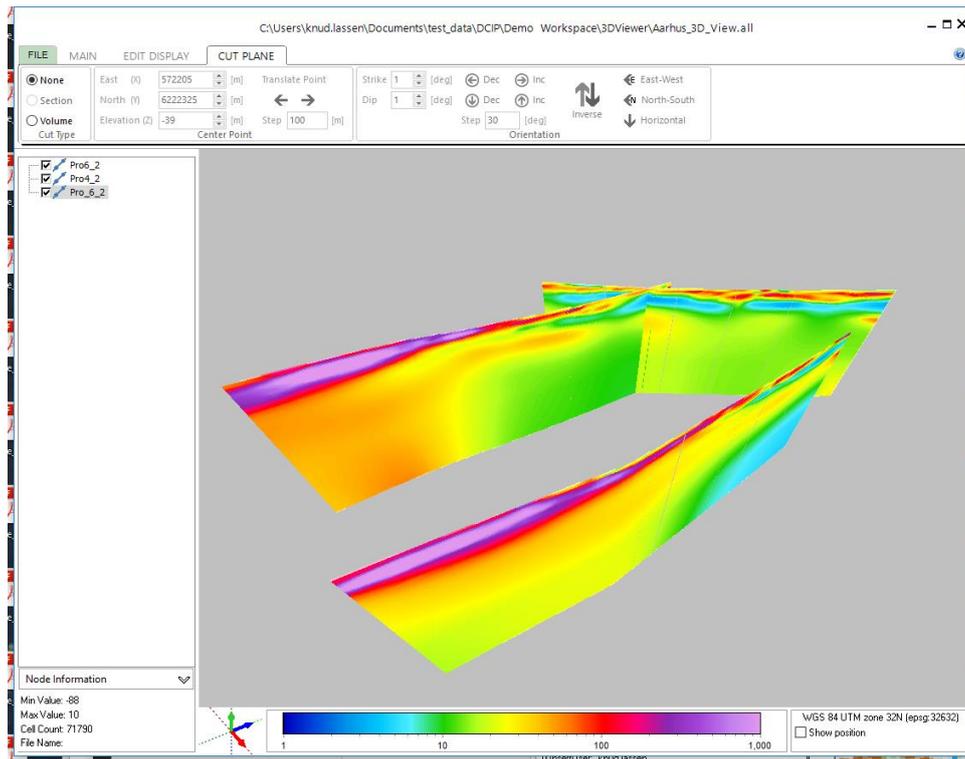
To create a new 3D view, navigate to the “3D” ribbon and select “Create”:



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:



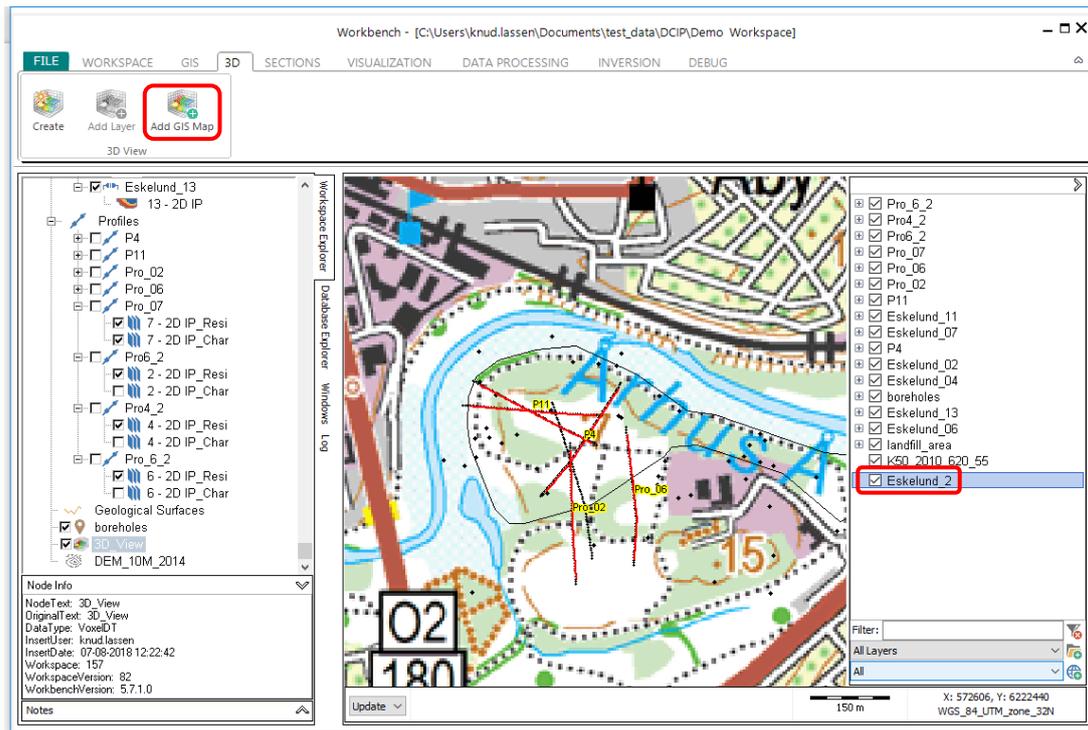
Navigating in the 3D viewer

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:

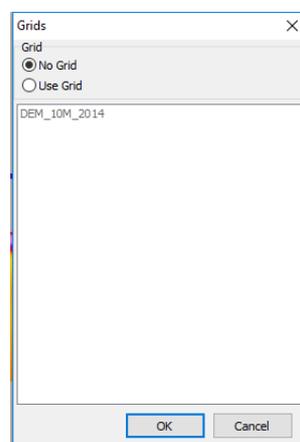


Location referenced images can also be imported directly to the 3D viewer by pressing “File->Load file”.

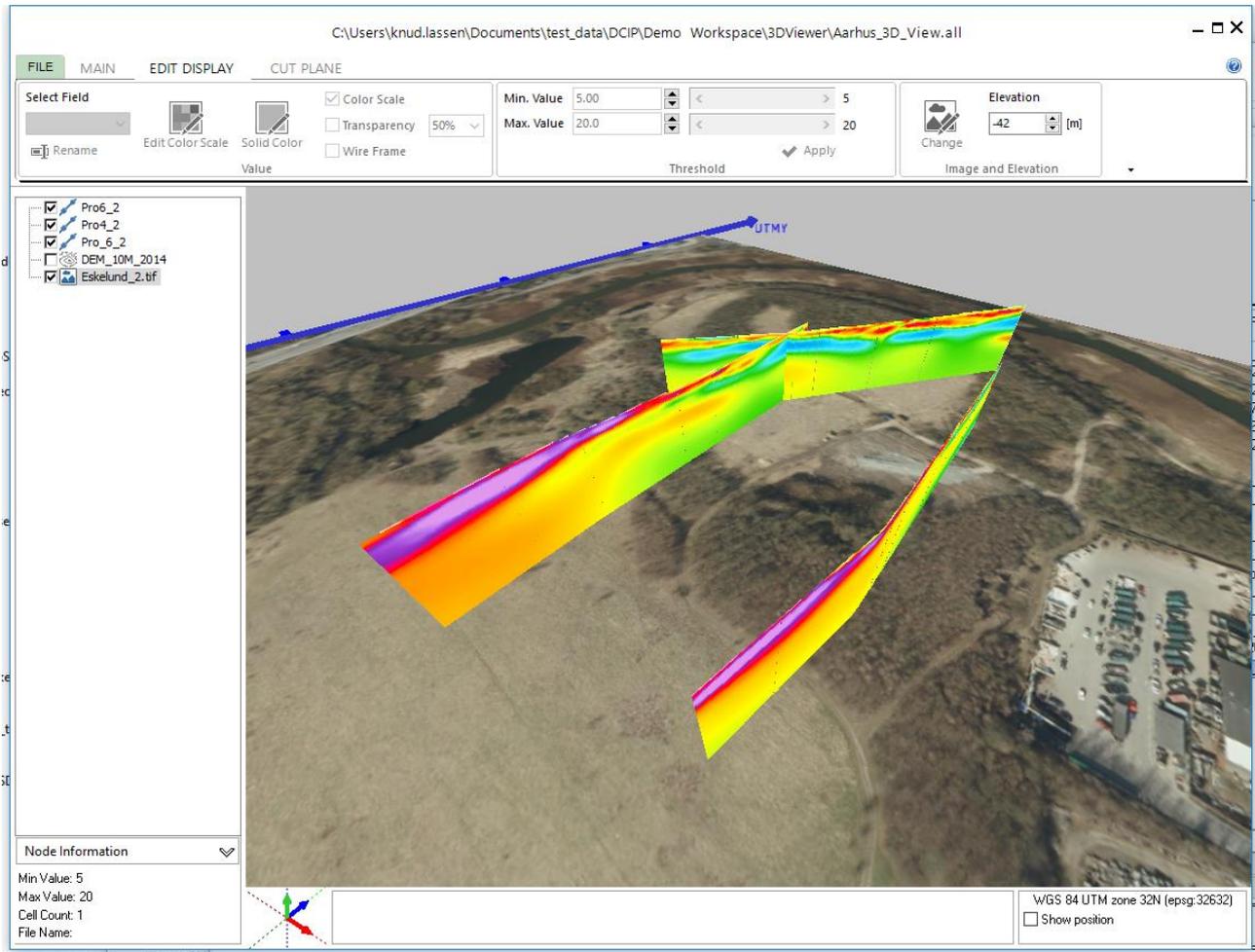
The other type of images are those created from gridded geophysical themes or gridded geological surfaces, these can be highlighted in the workspace explorer and added to the 3D view in the same ways as sections.

For both types of images, it is possible 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:

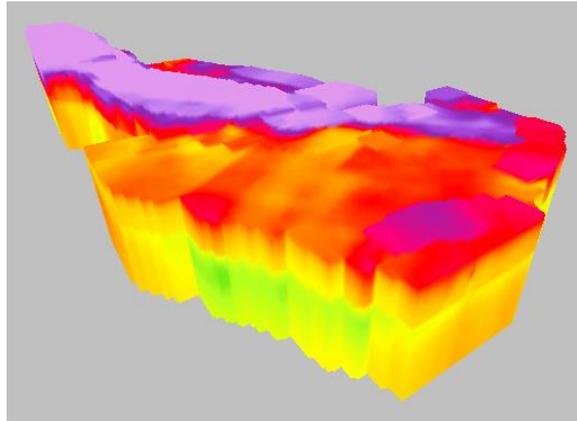


Cut planes, thresholding and 3D grids

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

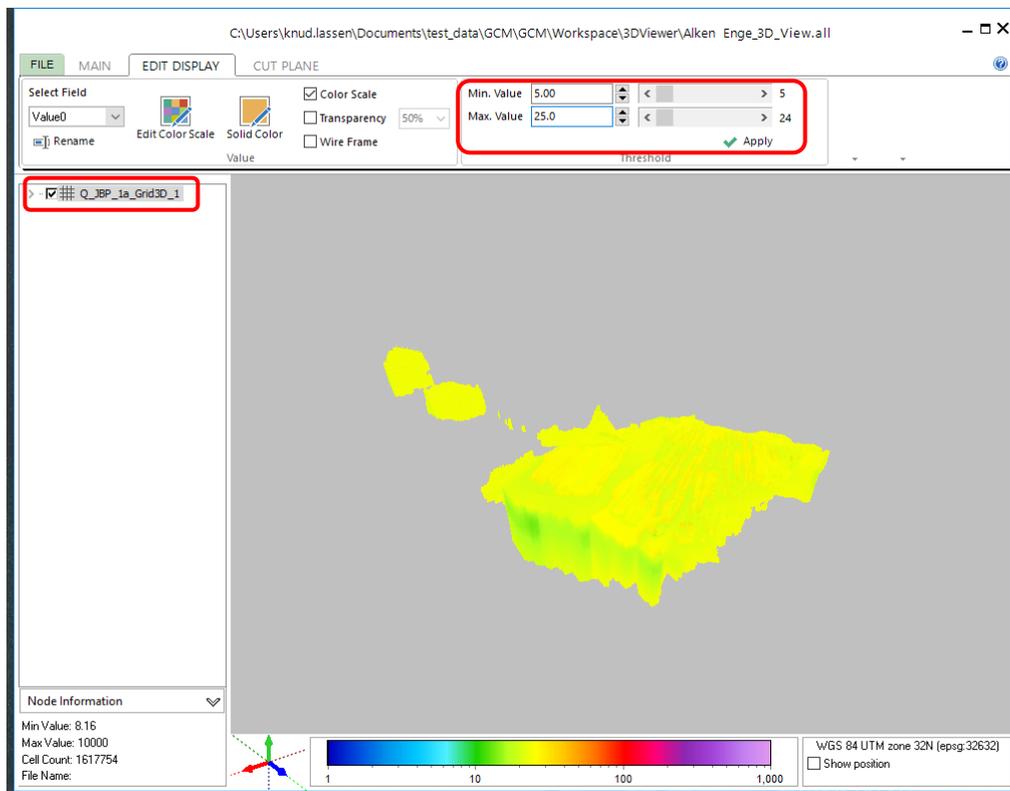
3D grids are added as all other data types by highlighting in the workspace manager and selecting “Add Layer”, they can also be loaded in directly as VTK files e.g. from external inversion or modeling software e.g. Res3DInv.

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:

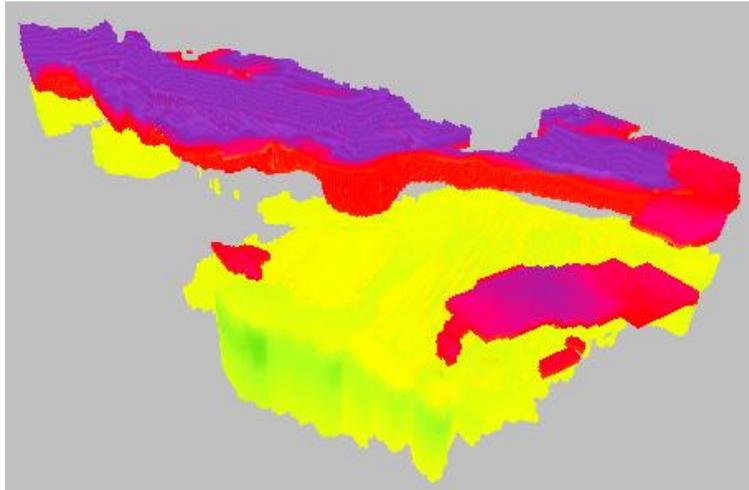


Thresholding

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:

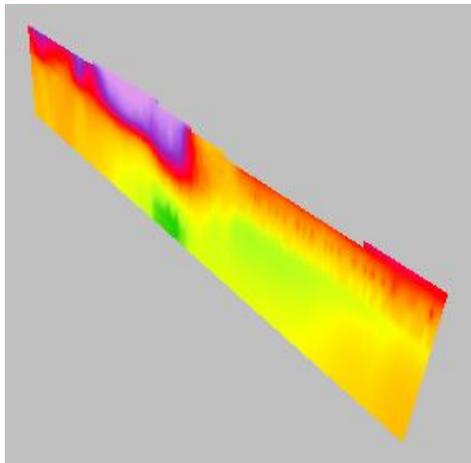


Cut planes

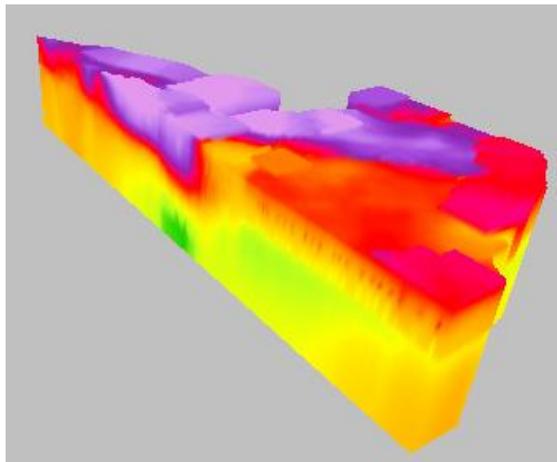
Cut planes makes it possible to make cuts through data to show relevant regions, they are found on the “CUT PLANE” ribbon:



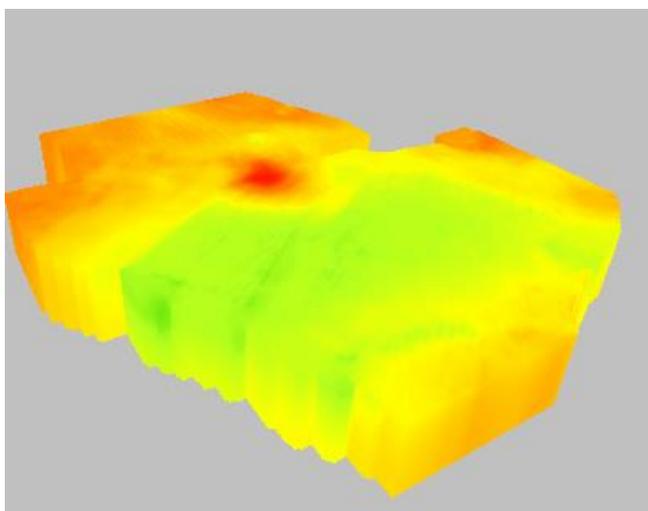
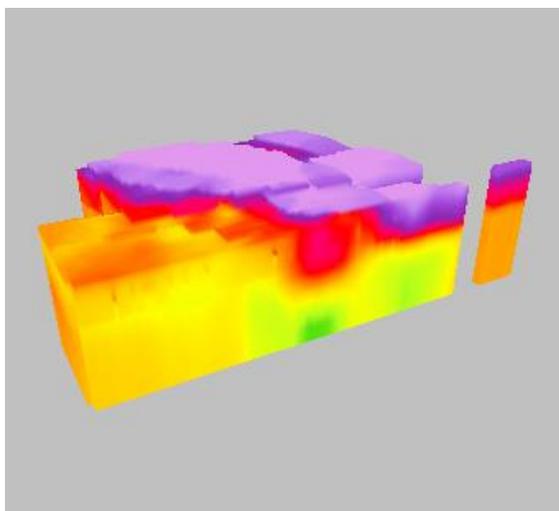
The default setting is “None”, there are two additional settings, namely “Section”, showing a slice through data:



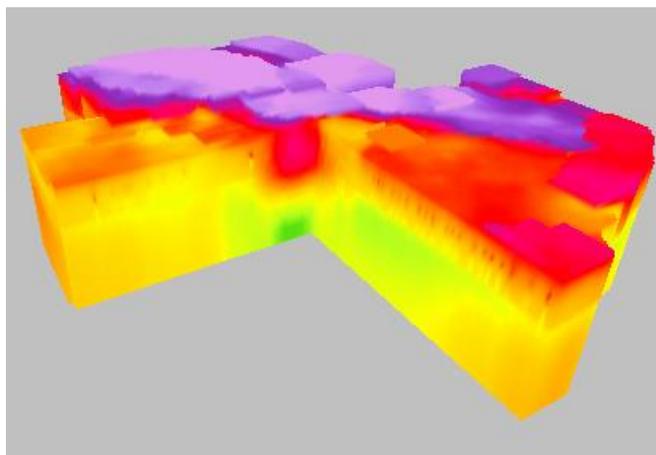
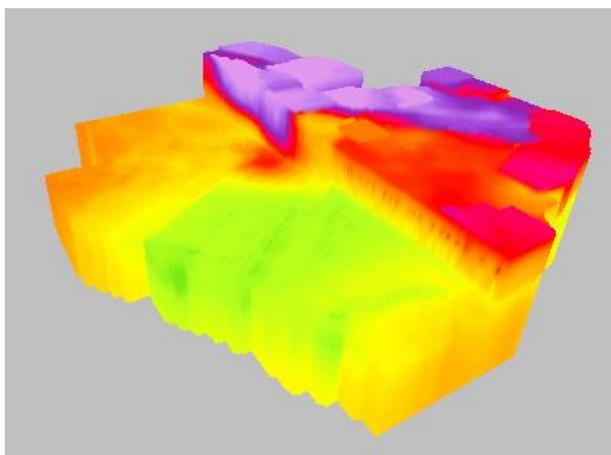
And “Volume” that cuts out part of the data:



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.

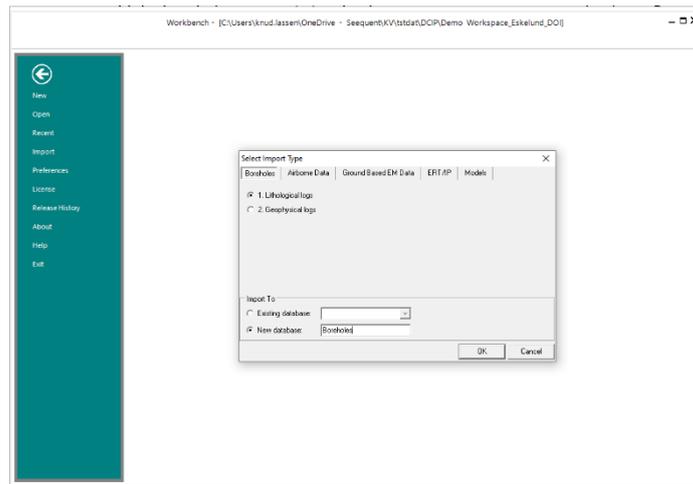
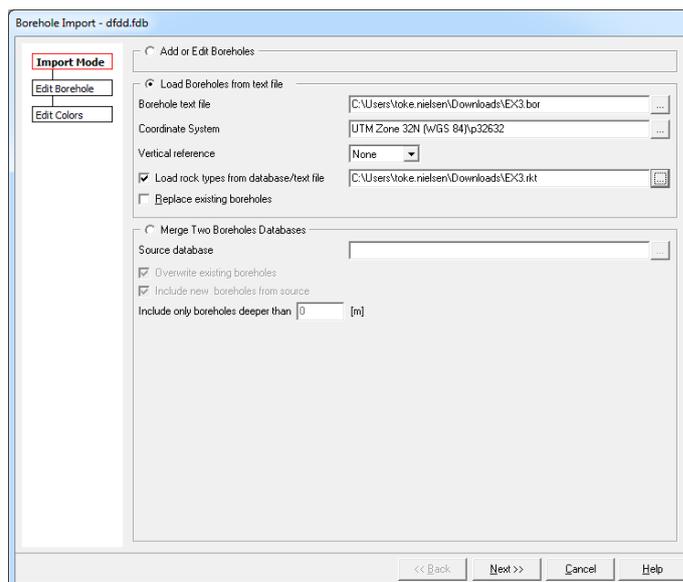
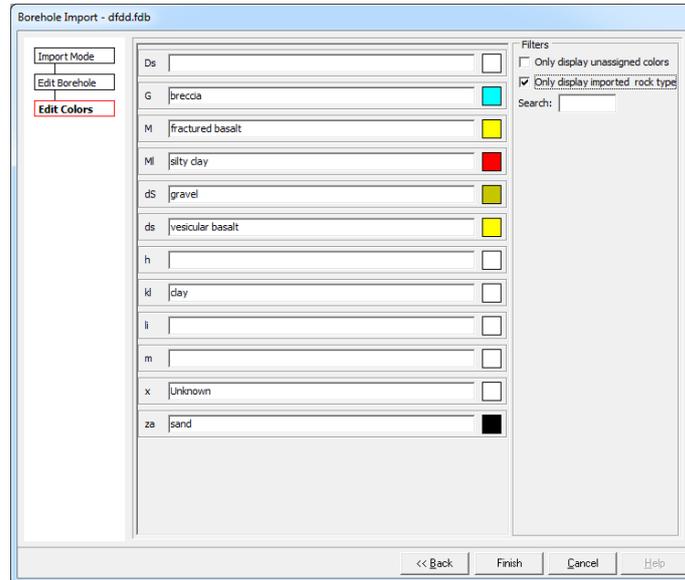


Figure 9

- 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 .rkt legend file exist, tick the “Load rock types from database/text file” and load the rkt file. Press “Next”.



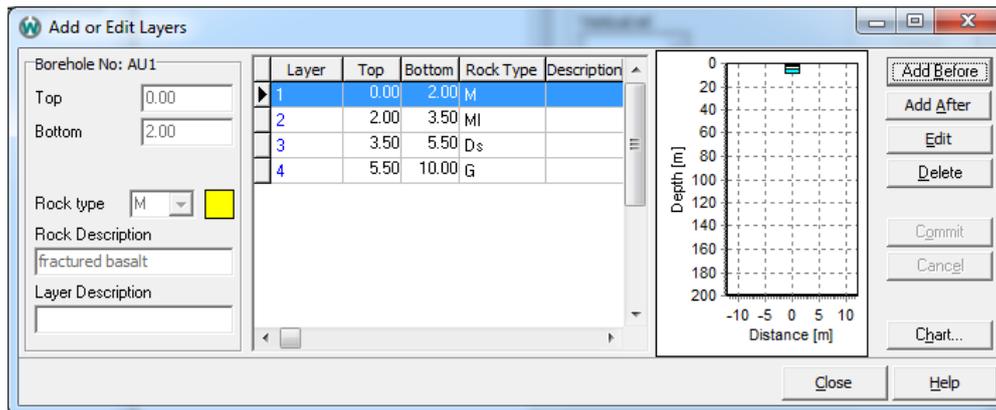
If the .bor file used does not contain the columns RGB and RockDescription, after pressing “Next”, no description and the same white color are associated to each rock type (Figure 3). It is possible to use the editing window in Figure 3 to manually modify the color (by left click on the squares on the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

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 to 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.