Aarhus Workbench hands-on course

-

Processing and inversion of VTEM data

PROCESSING AND INVERSION OF VTEM DATA

Exercise 1: Data import

The following exercise demonstrates how to import VTEM data to the database in Aarhus Workbench.

Exercise 1.1: Create an Aarhus Workbench workspace

Open Aarhus Workbench and start creating a workspace by pressing **New** to open the wizard. Fill out the following:

- Workspace **folder** path where the workspace is saved.
- **Workspace name**
- **User name**
- **GIS map node name**
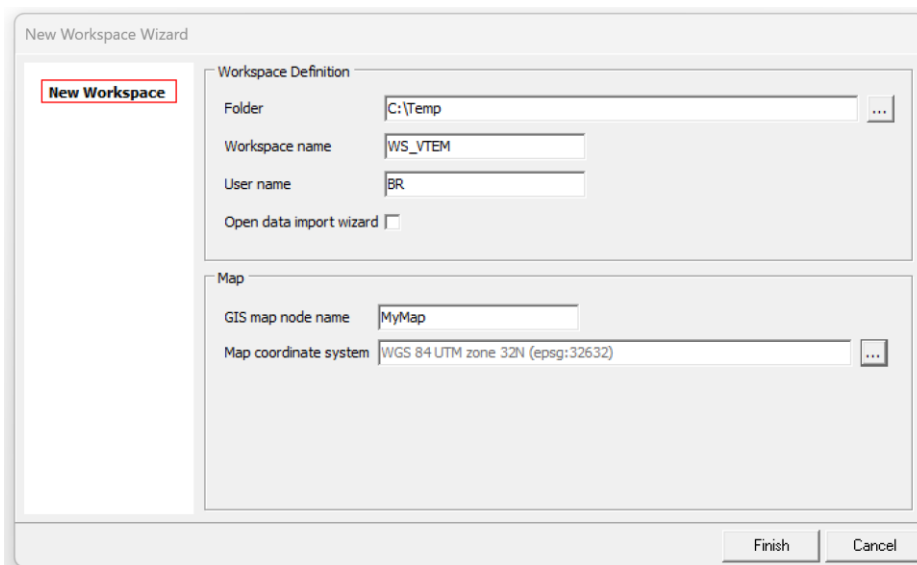- **Map coordinate system** for the GIS map.

Figure 1.1: New Workspace Wizard.

Exercise 1.2: Add maps to the Workspace

In this exercise we want to add different external maps to the workspace.

- In the bottom right of the GIS layer manager you can press the open icon to load maps into the Workspace.
- Try adding a few different layers. This can be background maps, satellite imagery and layers with power lines in the area, and so on. Often these will be in the form of tif or shape files.
- Try changing the order of the loaded maps. So that the background map for instance is behind the power lines. This can be done by holding down the mouse for a second and then dragging the map to the bottom in the GIS layer manager. It can also be done by right clicking and using the **Send layer to front/back** options.

- In the bottom right of the GIS layer manager you can also add web map service (WMS) layers. Try adding the default suggested WMS layer.
- For now, only have the background map visible, but the other layers can become relevant later.

Exercise 1.3: Import of VTEM data

Import of data is done from the Database Explorer. Go to the **Database** ribbon and select **Import.** This is **Airborne Data** more exactly **Airborne TEM processing**, so select that importer. Set I**mport To** to point to a new database and give it a name. This open the import wizard. Select the data type **VTEM** and move on to System Setup.

System Setup and Database Definition

Fill out the following:

- **Dataset Name**
- **Dataset Coordinate System** which is the map projection that the dataset will be saved to. It can be different from the projection used for the GIS, but there is rarely any reason for that to be the case.
- Point to the **System Setup File** (i.e. geometry file).
- The remaining data processing information including height System under **Labels** only serve as labels in the database. They are useful for users in Denmark where data is uploaded to a central database, for others they can safely be ignored.

Press **Import** to finish the first step of the import. A dataset with the relevant instrument information has now been created.
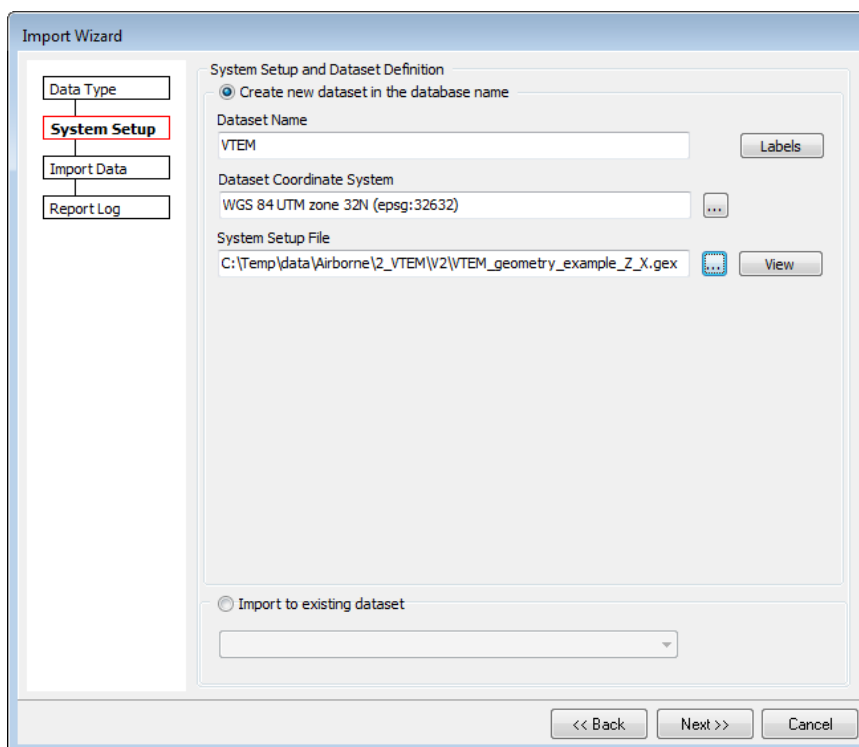

Figure 1.3a: VTEM Data import – System Setup page

Import Data

Import the VTEM data:

- Point to the **xyz file** with the data.
- Point to the **xyz format file** with the information about how the data from xyz file should be imported. The file can be edited by hand or using the alc file editor (see F1 help for details). Make sure so to clear the path if you want to create a new file from scratch.
- The date and time can be imported from the xyz file using columns described in the xyz file and with a date and time format set in the **read from file** option. Else the date and time can be set using the **define start time** option with a somewhat arbitrary start time and a sounding distance appropriate for the data.
- Fill out the voltage unit and the data normalization. If the data has not been normalized by the current, set the transmitter peak current to be read from the data file. Else use a nominal value of 1.
- There is an option to do decimation on the data and skip some of the data during the import.

Think about your import strategy beforehand. In most cases it is advisable to import all data in one dataset and divide them into flights or flight days when you start processing the data. It can however sometimes be necessary to do partial import and add more raw data later.

After the import, check the log for the import to see if there is anything of concern there. It can later be found under log in workspace manager

Figure 1.3b: VTEM Data Import – Import Data page

Importing more VTEM Data to an existing dataset

Try importing additional VTEM data. This time set **Import to** to point to the existing database and use the radio button on the **System Setup** page to **Import to existing dataset**. If you don't use real times, make sure to adjust the times to avoid importing to the same time interval.

Exercise 2: Setup of processing

This exercise introduces the processing system.

Automatic processing involves automatic filtering, averaging, etc. of data. We will run an automatic processing of voltage data, but we will not go into details with it until a later exercise. The result of the automatic processing will later be inspected using **Views**, which will be introduced in Exercise 2.2.
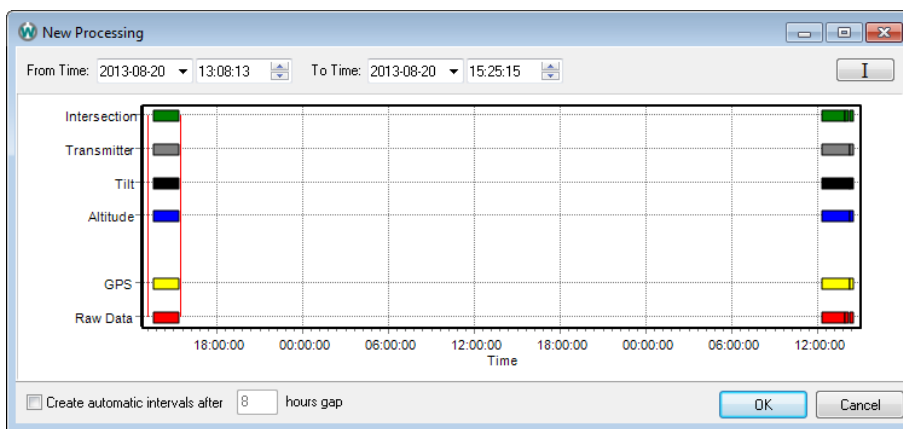

Figure 2.1a: The new Processing window where the time interval of the processing can be specified. Colorized bars show the distribution of different data types: Red bar: Voltage data, yellow bar: GPS data, blue bar: Altitude data, black bar: Tilt data, grey bar: Transmitter data and green bar the intersection between these.

Exercise 2.1: Initializing an automatic processing

Initialize the processing of the VTEM data by following these steps:

- Go to the **Data Processing** ribbon and select (Create New) **Data**. On the next form select the database and the **Airborne TEM** processing type. This opens another form where data belonging to the dataset in the database can be selected.
- Select the dataset to be use and enter a name for the processing. This creates a data node, which is the link to the dataset in the **Database Explorer.**
- Specify which time interval to work on from the **New Processing** window (figure 2.1a). Use the fields **From Time** and **To Time** or use the mouse. It is possible and easy to auto create intervals, but if more data will be added later, it is important to be familiar with the other options. In this case make a processing node for the first day. It can often be practical to limit the size of such nodes to one full day of production.
- When the time interval is specified the **TEM Processing** window (figure 2.1b) is opened and settings for the automatic processing can be entered. Just use the default settings for now.
- Press **OK**. This creates a processing node as a subordinate of the data node.

- Once the data node has been created, additional processing nodes under the same data node can be made by selecting the data node and selecting **Process Data** from the **Data Processing** ribbon. If more data is imported to the same dataset later, the processing nodes should always be made like this and not like it was done earlier creating a new data node with new processing nodes.

If you have a DEM for the area in surfer format, go to the **Workspace** ribbon and select **Add Elevation Model** to create a DEM node in the Workspace with the DEM. There is an option to cut the DEM down to the relevant part before it is added as a node. Ones the node is created, and this can take a while, it can be used to create an image. It is also now possible to point to this DEM node by selecting a data or processing node and then using **Add** (Elevation Model) from the **Data Processing** ribbon to update those nodes. The DEM must be added again when creating new nodes or if the whole processing is run again.
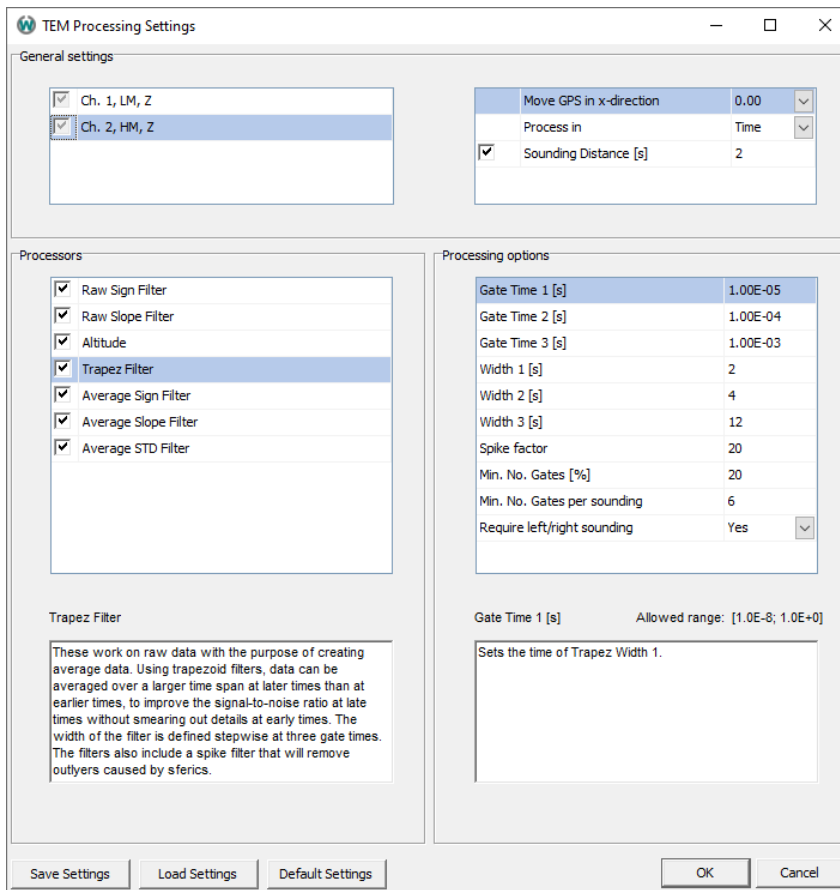

Figure 2.1b: The TEM Processing window. This window is the control center for the automatic processing. For now the defaults are fine, we will return this in a later exercise. On the left there is a list with the different channels of the data. Each selected channel will have its own list of Processor and Processor options. The blue background color is used to indicate what has been selected. Check the box in front of the channel to process that channel when pressing ok. The first time data is processed all channel must be processed (hence the greyed out boxes), subsequently it is possible to only processed the checked channels.

Exercise 2.2: Introduction to the manual processing system.

We will now try experimenting with the processing system to get acquainted with it.

- Tick on the Processing node in the **Workspace Explorer** to see the data positions on the GIS map**.** Confirm that they end up roughly where they should.
- Select the **Processing node** and click on **Show and Edit** on the **Data Processing** ribbon. This opens a data view in **Views** (figure 2.2).
- Move the mouse over the plot form and see how the helicopter position is displayed on the GIS map.
- Try zooming in on the dot showing the helicopter position on the GIS map. Move the mouse on the data plot again. Then try the same with **Auto Center Map** on the **View** ribbon unchecked.
- Now zoom out and try using **Find Nearest** from the **GIS** ribbon to click near some data positions on the GIS map while having the data view open. What happened?
- Go to the **Edit** ribbon and click on the **Buffer** button. This opens a window where one can click to place the buffer. When processing most of the navigation will be done using the arrow buttons around the **Buffer** button. Try making the **Buffer Interval** larger in the settings list on the right under General and then use the forward arrow to update the data plot. Now try changing the **Zoom Level for STD** setting to be slightly larger. What happened?
- Just below the **Buffer** button there is a line number dropdown. Try using that and its arrows instead.
- Note that it always is the active plot, as indicated by the blue border, that moves. Try to enable **Sync Data Plots**. This will let the other data plots show the same time/distance intervals.
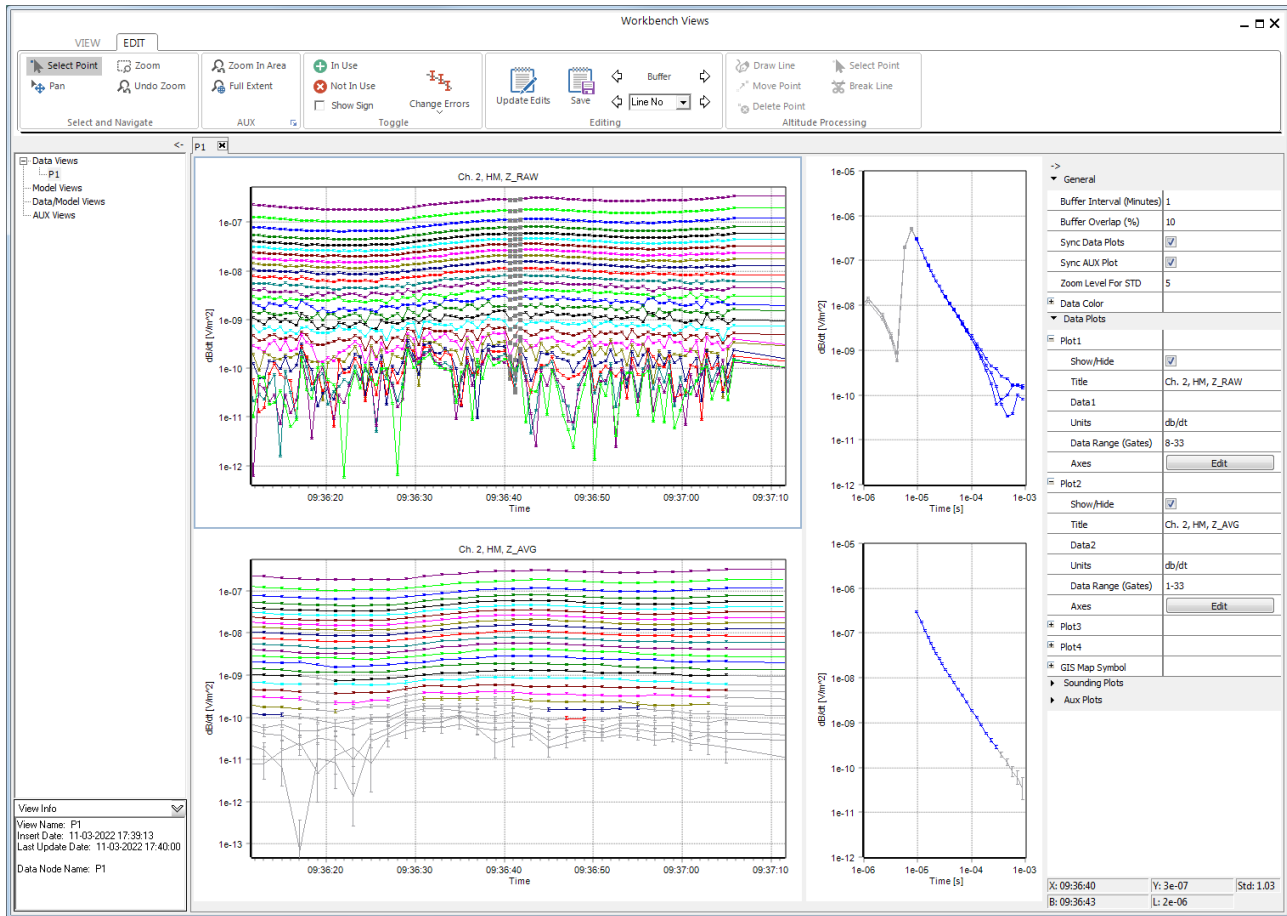
Figure 2.2: Views with a Data View open showing TEM data. The left side show the views that have been created. The center is showing data plots with high moment raw and average data as well as sounding plots corresponding to some selected positions near the center indicated with darker grey points. The data plots show dB/dt voltages of soundings plotted next to each other in recording order. Each colored gate corresponding to specific time gates of those soundings. The first colored gates correspond to the first blue gate visible on the sounding plot, and so on. The axis setup of the data plot and sounding plots are slightly different, otherwise the gates would be at the exact same level. The light grey gates on the data plots and sounding plots have been disabled and the early light grey gates on the sounding plot have been hidden on the data plot. On the right we have the different settings for what is being plotted. It is usually not necessary to only show 1 minute of data, that was done to make it fit on this page, but you need to be able to see roughly this level of details. The Transmitter Altitude can be plotted on the Aux Plot.

Exercise 4: The automatic voltage data processing

In this exercise we will work with the automatic voltage data processing. It is split into 3 parts as we take on each of the 3 main groups of filters: The **Raw** filters, the **Trapez** filters, and the **Average** filters. Even within these groups it can be helpful to experiment with the different filters individually, to get a better understanding of what each filter does. This exercise is more about that understanding than about finding the perfect filter setup.

At the end of the exercise we will look at how your data determines which filters to use, as it may not always be advisable to use them all.

7

For the purpose of these exercises we will be using the default **Process in** setting of **Time** rather than **Distance**.

Useful keyboard shortcuts during processing are shown below.

Shortcut	Function
Hold Shift while selecting data	Show the sounding corresponding to these positions on sounding plot.
Hold Shift and Ctrl while selecting data	Select the gates corresponding to these positions across all visible data plots.
a	Toggles data points on.
q	Toggles data points off.
Alt # (# denotes keys 1-9)	Adds 5, 10, 15%,... uncertainty to selected data points, 0 removes it again.
Ctrl + S	Update and save the processing.
Ctrl + U	Update the average data

Table 4. Useful processing shortcuts.

Exercise 4.2: The Raw filters

In this exercise we will do experiments with the **Raw** filters of the TEM processing. The **Raw** filters are there to take out capacitive couplings or at least enough of them so that they are easily spotted in the manual processing and can be removed there.

First a bit of preparation.

- Open the data plot in **Views** by selecting the processing node and clicking on **Show and Edit.**
- Show the raw and average data using the **Show/Hide** buttons for the different plots on the right side.
- Set the **Buffer Interval** to 2-3 minutes of voltage data. We need to be able to see the variation in the data from one sounding to the next. How much data you can comfortably show of course also depend how much room you have for Views on your screen.
- With some imports it can be necessary to adjust the **Data Range (Gates)** for the raw data to not show any early auto disabled (grey) gates.
- Look at the raw data. Has data been removed by the automatic filters? Find a place where you both have data that should be kept and data that should be removed.
- The filters work on individual soundings so we will also need to evaluate using sounding plots. Hold down shift and then select part of one or more soundings on the data plot. This is used to plot the soundings at the selected positions on the sounding plots.
- The axis of the sounding plots can be adjusted to show a fixed range, rather than an automatic range based on the data. This is often preferable when doing a lot of comparisons. To change

the axis either use the **Axis Edit** setting on the right under Data Plots or right click on the sounding plot and use **Axis Properties**.

Now we are ready to return to the TEM processing.

- Open the **TEM Processing** form again by selecting the processing node and clicking on **Process data**.
- Try only having either the **Raw Sign** or the **Raw Slope** filter active when you run the processing. Then go back to the sounding plot and evaluate. Which one appears to have the greatest impact on the data here when using the default values?

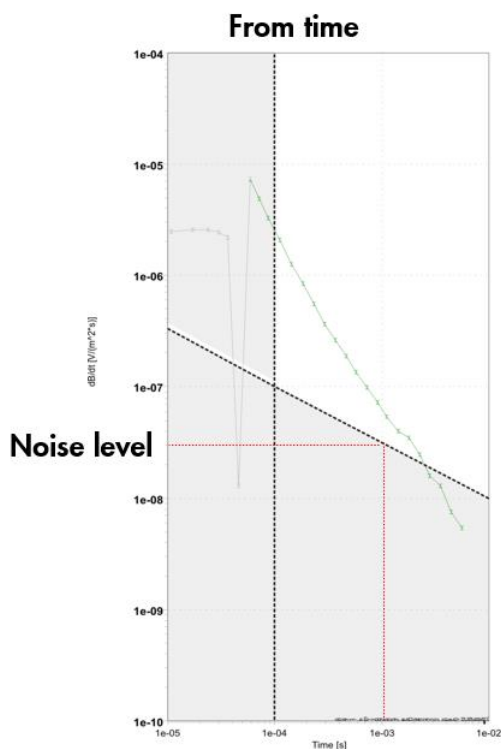To really understand these filters we need to look closer.

Figure 4.1: The Raw filters. Each gate is checked to see if it triggers the condition specified by the filter, either a sign change or a second order slope change above or below the threshold value. Starting at the earliest time moving towards the latest time, but excluding the gates in the darkened area described by the From time and the Noise level. The Noise level line is defined by its value at 1ms. Ones the condition trigger the filter will remove all later gates and it can be set to back step and disable a number of earlier gates as well.

- We want to be sure that the **Raw** filter trigger because of capacitive couplings, so we need to be able to exclude the false positives triggers we otherwise would get from the noise at late times.
- When this filter was made some systems still measured the noise level, but this is very rarely if ever done anymore. We could still try to estimate the noise level by looking at sounding plots, but in practice it is often easier to approach the noise level with a bit of trial and error.

- Set your filters to have a back step of a lot, so that when the filter triggers on the noise, it removes all the data on that sounding. Then try adjusting the noise level up or down until it doesn't trigger on the noise, but only where you have couplings. A typical value might be around the default 5e-7. It is better to enter a little too high a value than to have the filters remove noise as couplings.

With that sorted we can start playing with the other options.

- Go through the remaining **Raw Sign** filter options one by one, to see the effects. Note that you can use **Show Sign** in **Views** to actually see the signs of the data. If there is little to work with just skip ahead and try the same things with the **Raw Slope** instead. Try using Back step at 0 to see exactly where the filter triggers. Try changing From time.
- Find a set of settings for the **Raw Sign** filters which seem to give good results. Remember not to make the filter remove too much data and keep in mind that this doesn't have to be perfect, this is about making the manual processing easier and faster.
- Try the same for the **Raw Slope** filter. Keep the minimum and maximum values of the second order slope change symmetrical. The smaller the value, the easier it is for the filter to trigger.

Note that you can save your processing settings to a folder.

Exercise 4.2: The Trapez filters

With the capacitive couplings now at least partially removed, it is time to experiment with the **Trapez filters**.

Note the option to **Keep Manual processing of Raw Data**. If this is ticked any processing in the raw, including manual processing, will be kept even when the processing is rerun.

- Open the data view in **Views** by selecting the processing node and clicking on **Show and Edit.**
- Show the raw and average data. Show a few raw and average soundings in sounding plots. Take a screen dump and paste it in e.g. a Word document also noting approximately which soundings were selected.
- Open the **TEM Processing**.
- Try decreasing or increasing (significantly) the **Trapez Width 3**, run the processing and see what happens to the noise level.

The purpose of the trapez filters is to improve the signal-to-noise ratio at late times, but also to avoid smearing of likely geological features in early time.
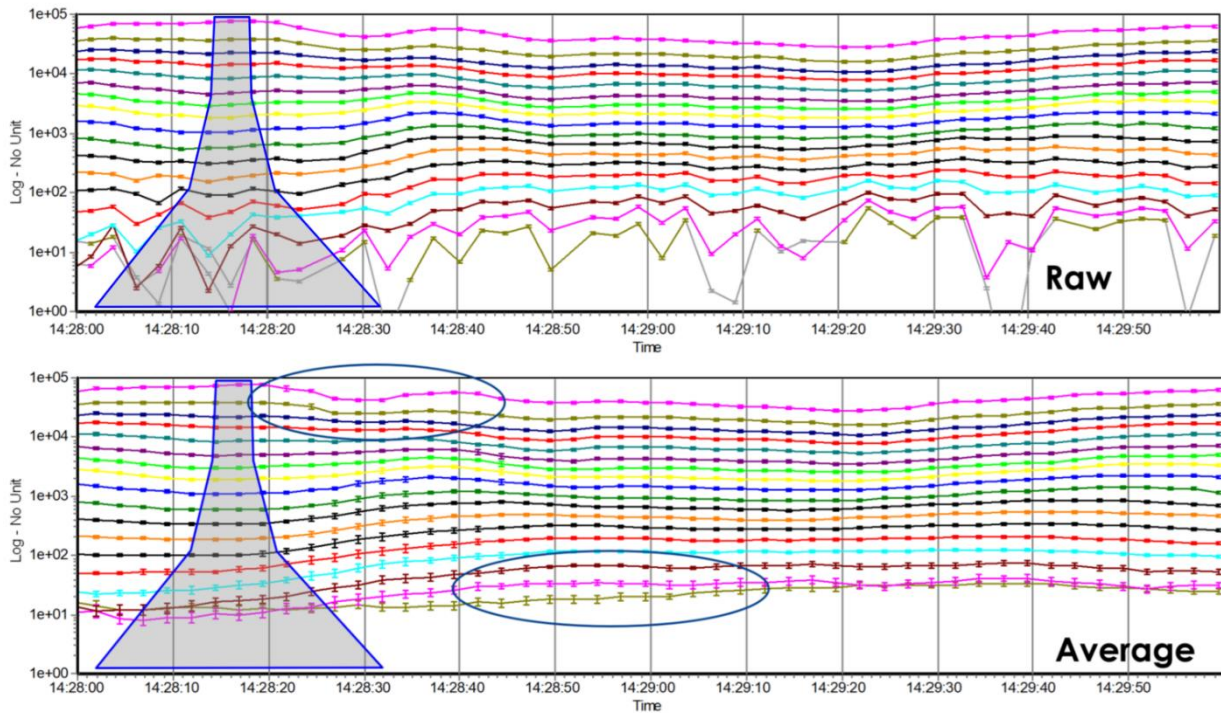
Figure 4.2a: The Trapez filters. Using a trapezoidal shaped running mean in the stacking of the raw gates to average gates allow us improve the signal-to-noise ratio at late times, without smearing likely geological features in the early times. Such a shape also to mimic the expected footprint of the method.

How often a sounding is made is set with the **Trapez Sounding Distance**. This one is based on the flight speed, and usually set so that there will be one average sounding every 20-30 m. This roughly correspond to the size of the footprint of the system at the surface. Any closer than that and we would increasingly end up inverting the same data in neighboring soundings.

- Open the data view in **Views** by selecting the processing node and clicking on **Show and Edit.**
- Show the **Speed** on the aux plot. If the data was imported with real times, this can be helpful in picking the **Trapez Sounding Distance.**

Normally the **Trapez Sounding Distance** is set a little higher than what the average Speed to account for differences throughout the survey. And then **Trapez Width 1** is made yet a bit wider.

When using narrow widths, make sure to set **Trapez Require left/right sound** to **off**, as it will otherwise end up removing some early average gates.

For **Trapez Width 2** and higher it is all about avoiding smearing at early times while improving the signal-to-noise ratio at late times.

It is recommended to leave the default **Trapez Gate Times** unchanged and focus on the **Trapez Widths**. It is possible to tailor the **Trapez Gate Times** to the first and last usable gates, but it is usually not worth the effort. It can however be a good idea to compare the times with a sounding plot, just to have an idea about where the stacking changes.

- Show the raw and average data. Look for an area with fast level changes that could be due to geological features (depends on the area). Are they preserved in the averages? Show a few raw

and average soundings in sounding plots. Try to use screen shots to compare. Make sure to note which soundings where selected.
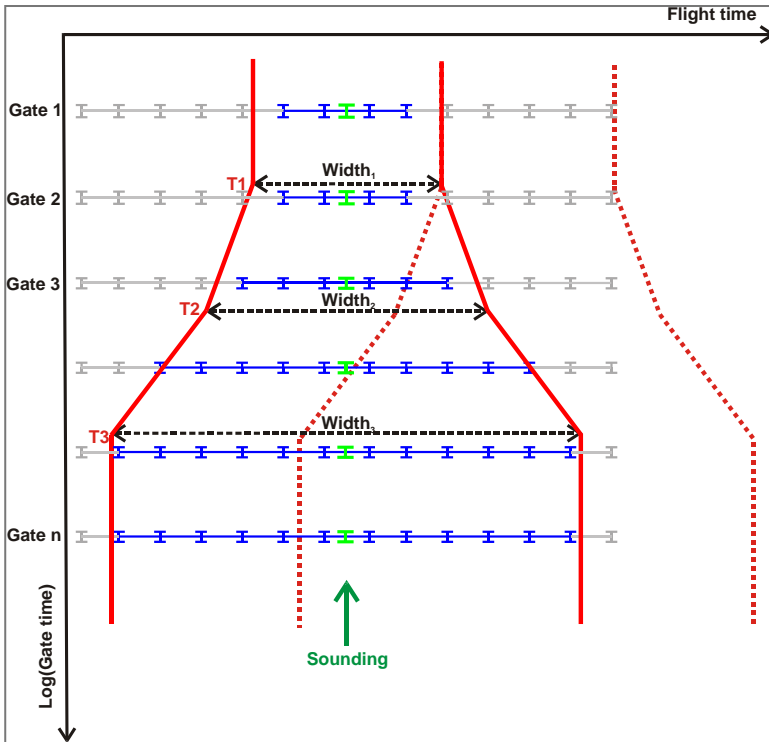

Figure 4.2b: Detailed trapez filter setup. Again showing the soundings side by side, the flight time along the x axis and the gate times along the y axis. The raw gates are grey, or blue if they are inside the trapez. The red trapez is created from the trapez gate times and the trapez widths in the TEM processing. For each sounding distance we try to create an average sounding, the green gates, based on the blue gates inside the trapez, but subject to the spike filter, some minimum requirements and possible the require left/right setting that only forms an average gate, if there are raw gates on both sides of it. Then we move one sounding distance down the line and create the next trapez, the dotted red line.

Note the option to **Keep Manual processing of Raw Data**. If this is ticked any processing in the raw, including manual processing, will be kept even when the processing is rerun.

Exercise 4.4: The Ave filters

In this exercise we will do experiments with the **Ave** filters of the voltage data processor. The **Ave** filters are a simpler version of the **Raw** filters, but they have a different purpose. They are not there to look for couplings. All coupling should have been removed in the raw data. The **Ave** filers are there to catch the point where the averaged data goes below the noise level, where it goes from having a data dominated $t^{-5/2}$ curve to noise dominated $t^{-1/2}$ curve.

- Show the raw and average data. Find an area where the late times look noisy. Show a few raw soundings in a sounding plots as dB/dt to see if the sounding curve change can be observed.

- Find a set of settings for the **Ave Sign** and **Ave Slope** filters that take out the noise where you think it should be done.

It is always important to use the **Ave Sign** filters to prevent negative data from being send through to the inversion.

- The more recently added **Ave STD** filter can also be used.
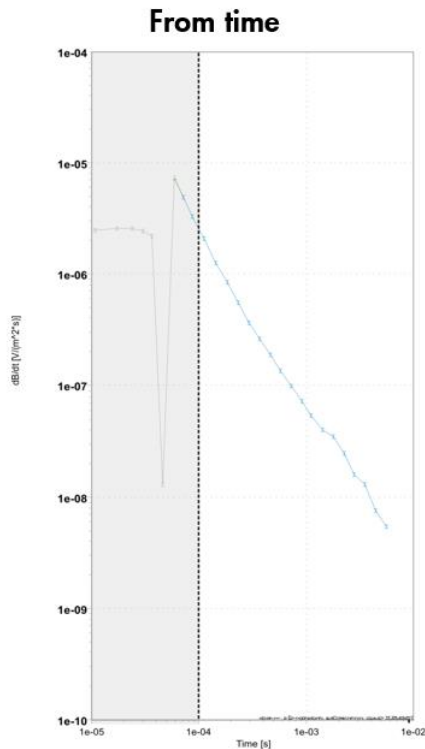

Figure 4.3: The Average filters. Each gate is checked to see if it triggers the condition specified by the filter, a sign change, a second order slope change above or below the threshold value, or a STD above a threshold value.. Starting at the earliest time moving towards the latest time, but excluding the gates in the darkened area described by the From time. Ones the condition trigger the filter will remove all later gates and it can be set to back step and disable a number of earlier gates as well.

Different data, different approaches.

We have now been through the most important automatic filter options. The specific filters may need to be tuned for a given survey, but before doing any of that, it is worth considering what filters we even want to use here.

If the data already has been preprocessed, we may not want to do any additional processing. This can be done by disabling the Sounding Distance and the Trapez filters. In this case the average data is simply a copy of the raw data.

If there is some room for improvement in the preprocessing, we can decide to use the Raw filters to look for capacitive coupling in the raw data and the Average filters to remove the noise in the average data.

We can also decide to apply the Trapez filters to additionally improve the signal-to-noise ratio while at the same time trying not to smear the early time data. It depends on what already has been done to the data we have been given.

Finally we can decide that we want to use a different Sounding Distance. This also requires the use of the Trapez filters. There can be different reasons for this, including that the data is so densely sampled that it hardly makes sense to keep the original Sounding Distance. In that case however we would risk smearing the data with the Trapez filters, so perhaps it would then be better to go back to the import and decimate the data to a more reasonable Sounding Distance.

The settings for the Sounding Distance and the Trapez filter can be set either in **Time** or **Distance** by switching the **Process in** setting. Each approach has advantages and disadvantages.

Exercise 5: Manual Voltage Data Processing
When the automatic voltage processing has been adequately set up and run it is time to move on to the manual voltage processing.

Manual voltage data processing can be divided into two steps. First the removal of coupled data. This is done from raw data as it is vital to remove these before the final data averaging. Second the noise processing is carried out. This is done on averaged data as it is important to obtain a better signal-to-noise ratio before this step.

Exercise 5.1: Raw and average manual processing
In this exercise we will remove couplings in the voltage data.

- Show raw and average data. Also show **Transmitted Altitude Edited** on the aux plot.
- Set the **Buffer Interval** to 2-3 minutes of voltage data. We need to be able to see the variation in the data from one sounding to the next. How much data you can comfortably show of course also depend how much room you have for Views on your screen.
- Now check that you are using the **Select Point** pointer, and then make a quick first impression evaluation of the data on the data plot. The automatic filters will often have a few suggestions for where you should look. It is not necessary to do the more time-consuming evaluation using sounding plots for this. Use the GIS to confirm potential couplings. Do not worry about details yet. Use the shortcuts from table 4, the q to disable and a to enable data shortcuts in particular will save you a lot of time.
- Press **Update Edits** and evaluate the result. Are the couplings still visible in the raw and the averaged data?
- Now go through potential trouble spot using sounding plots. This include sounding at the edge of the earlier quick removals. Did you remove enough? Hold down shift and then select part of one or more soundings.
- The sounding plots by default show dB/dt like the data plots. This is also most convenient when adjusting the automatic processing filters, but when evaluation potential coupling remnants, it is often best to shift to Rhoa (see figure 5.1a) to make the couplings stand out more clearly. This is done with the **Unit** setting on the right under Sounding Plots.
- The axis of the sounding plots can be adjusted using the settings file to show a fixed range, rather than an automatic range based on the data. This is often preferable when doing a lot of comparisons. To change the axis either use the **Axis Edit** setting on the right under Data Plots or right click on the sounding plot and use **Axis Properties**.

- Partially removed capacitive couplings are easy to spot on a sounding plot, but can often also be seen from the diverging or converging curves on the edit form near the edges of the partially removed coupling.
- Galvanic couplings are harder to spot, but pay attention to the changes in the signal level. Do the signal level changes correlate with the altitude changes? How about with potential coupling sources on the map? Keep in mind that signal level changes also can be due to changes in the subsurface.
- Press **Update Edits** and evaluate the result again. Repeat as needed. Move on to the next time interval and repeat a few times.
- Remember to **Save** the processing from time to time. Doing so also includes an **Update Edit** so there is no need to do both.
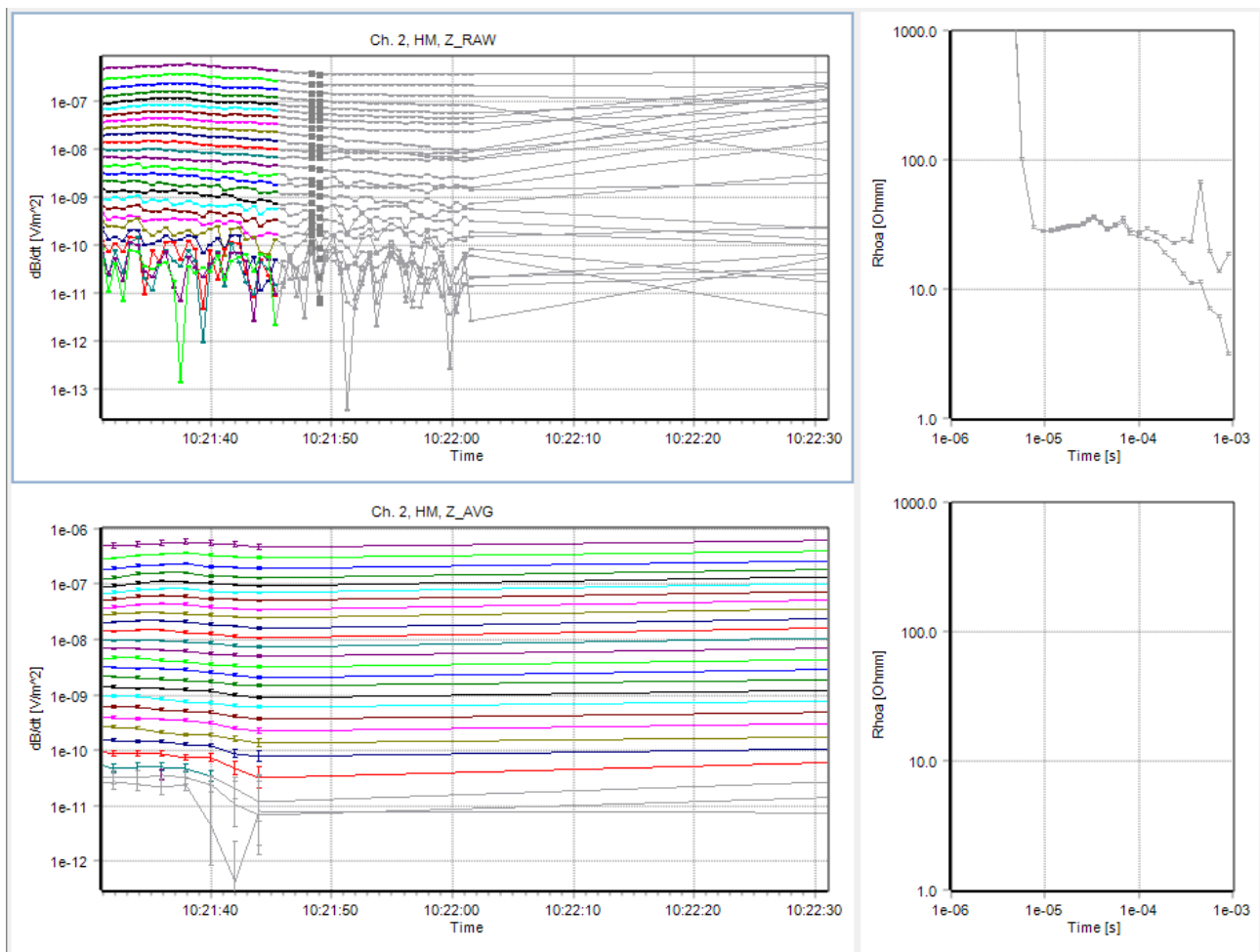

Figure 5.1a: Example of some coupled data, and how they have been removed. The Rhoa sounding plot clearly shows that the coupling still is present at the selected positions, even though the lines on the data plot at first glance might look fine. Looking closer one can see how the lines diverging or converging to form the bumps that we see more clearly on the Rhoa sounding plot.

In this second part of the exercise we will go back to the beginning and look at the noise.

- When the couplings have been removed in the raw data it is time to remove the noise in the averaged data. The automatic filters are quite capable at doing this, so in general one should spend rather more time checking if couplings have been correctly removed than checking if the noise data has been correctly removed.
- Note how far you have come and then use **Buffer** to go back to the beginning.
- Go through the soundings by selecting a distinct sounding or a group of soundings (see figure 5.1b) at a time and confirm that the automatic filters have removed the part that has gone below the noise level.
- Keep in mind that we average data to be able to use some of the data that appear as noise on the raw soundings. Remove or perhaps add uncertainty to noisy data points of the averaged data.
- Just as with couplings, it is often enough to look at the curves of the data plot and only use the sounding plots to confirm every now and again.
- Move on to the next time interval and repeat.
- When you get to time interval where you stopped and returned to the beginning try to continue doing the coupling first and the noise right after before going to the next time interval. Repeat until all data in the processing node has been evaluated or move on to the next exercise after a while.
- Try experimenting with the option to hold down Ctrl+Shift while selecting the raw data that should be disabled. This disabled the selection across all visible data plots, in this case making it possible to prevent averages in the interval.
- Try out the **Triangle** selection option as an alternative to the **Rectangle.**

Hints and tips
- Use F1 to open the wiki if you want to see the details for the settings on the right. We have only gone through the critical ones.
- Couplings should always be removed in the raw data, if they are kept in the raw data, they can affect more average data through the trapez filters.
- Noisy data should always be removed in the averaged data. If they are not kept in the raw data, they cannot be stacked and help improve the signal-to-noise ratio.
- Don't show more data than you can handle on the data plots, it depends on geology and experience, but 2-5 minutes of voltage data is usually about right.
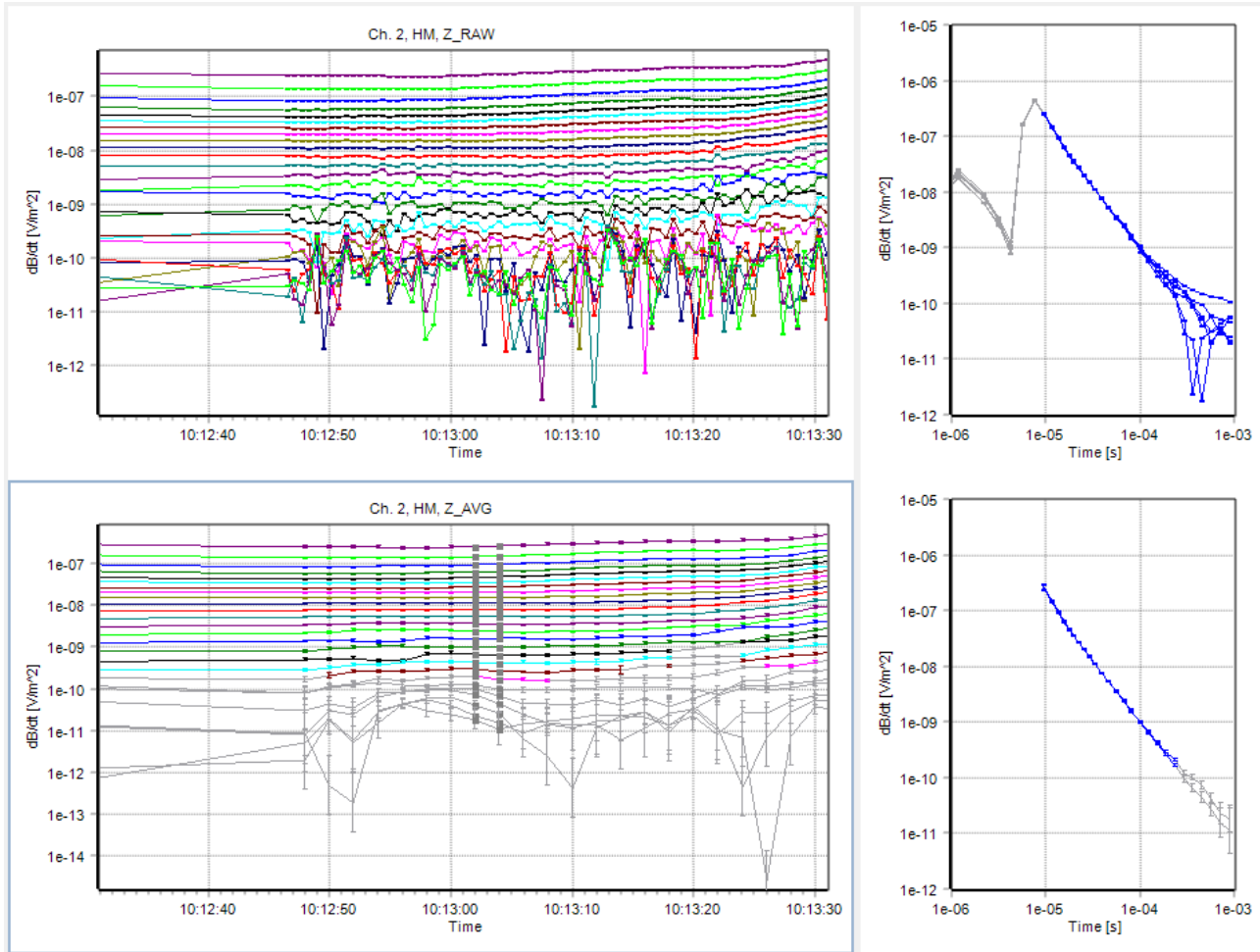
Figure 5.1b: Example of how the filters have removed noisy data. We can see how the raw data has been trapez averaged to improve the signal-to-noise ratio at late times, but also that there are limits to how far one can take this.

Exercise 6: Provisional Inversion

In this exercise we will look at inversion.

Exercise 6.1: Inversion parameters and provisional LCI

In this exercise we will run provisional laterally constrained inversions (LCI) of the data we have processed in the previous exercises. This will be used to make a post processing of the data. Later we will inspect the inversion result.

- Select the processing node and click on **LCI** on the **Inversion** ribbon.
- Now setup a small resistivity inversion. Make sure to check all the channels that should be inverted (usually just z component) and use **Set Time Interval** to invert the first 10 minutes of the processing node. Use **Get Entire Time Interval** to get the dates filled out and set the time interval.

- Use a smooth model. Try to edit the default model to consist of 20 layers with a first layer boundary in 3 meters and a last layer boundary in 300 meters. Is that layer distribution appropriate here? Use auto resistivity or set a uniform starting resistivity. Set the reference distance to your sounding distance. Consider what the effect would be if the vertical and lateral constraints are changes. Set the inversion to use as many CPU's as you can spare. The rest can be left alone for now. Hit **Run** to start the inversion.
- This launches the inversion through ABI in the background. This allows you to still use the Workbench although maybe a bit slower than before.
- While the inversion is running consider why we use a smooth inversion and not a layered inversion for this.
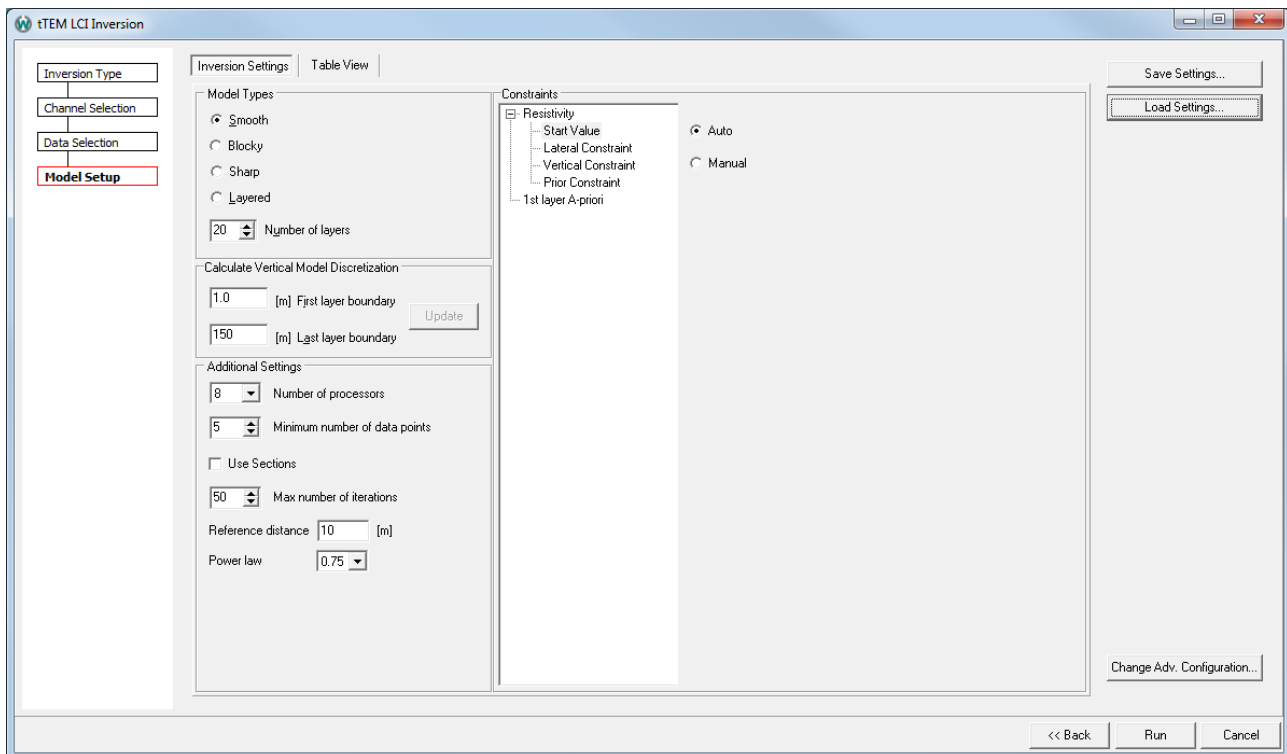

Figure 6.1: The last part of the inversion setup.

Exercise 6.2: Evaluation of inversion results, post processing
In this exercise we will take a first look at the inversion result of the provisional LCI.

- Close ABI when the inversion is done. Select the new inversion node in the workspace manager and click on **Show Result** under the **Inversion** ribbon. This opens a data/model view in **Views.**
- Here we have the models from the inversion showing together with the data that was used for the inversion. With the **Sync Data Plots** checked in the settings list on the right, they will line up and make it easy to adjust the processing where additional processing is needed.
- Before we start looking at the inversion result, we may need to make some slight adjustments to the setup. Most importantly for evaluation, we will want to show the data residual and how the inversion changed the altitude during the inversion. Use the **QC Plot Edit** option in the settings

on the right under Model Plots to add the **Data Residual** and both **Altitude** and **Altitude In-verted**. It is also possible to change the **Color Scale** or the **Model Display** with the width of the bars or how the model should be blinded below the depth of investigation.

- Now we can inspect the inversion result. Click on the models to see how the model fits the data on the sounding plot.
- It is also possible to go back to the **View** ribbon and open a **Line Plot** showing the model (in red) and the depth of investigation (in blue). And with **Model Parameter** one can see the exact model parameters as well as the starting model parameters. This can be quite useful at times, but for the most part it is the sounding plot we use to evaluate.
- Now we can go back to the **Edit** ribbon and use the **Buffer** or **Line numbers** to go through the inversion result.
- We mainly look for models with unusually high data residual or large changes between the starting altitude and the inverted altitude.
- Also look for suspicious changes in resistivity in particular near data gabs. These are usually the result of couplings that should have been removed at bit more.
- Remember to keep an eye on the GIS while doing this to get a better idea about where you should expect couplings.
- It may be necessary to do some additional manual processing after the first inversion. Often it is enough with some minor adjustments in the average data, but it is possible to switch the data plots using the **DataX** dropdowns in the settings list on the right under Data Plots and get full access to both the raw and average data.
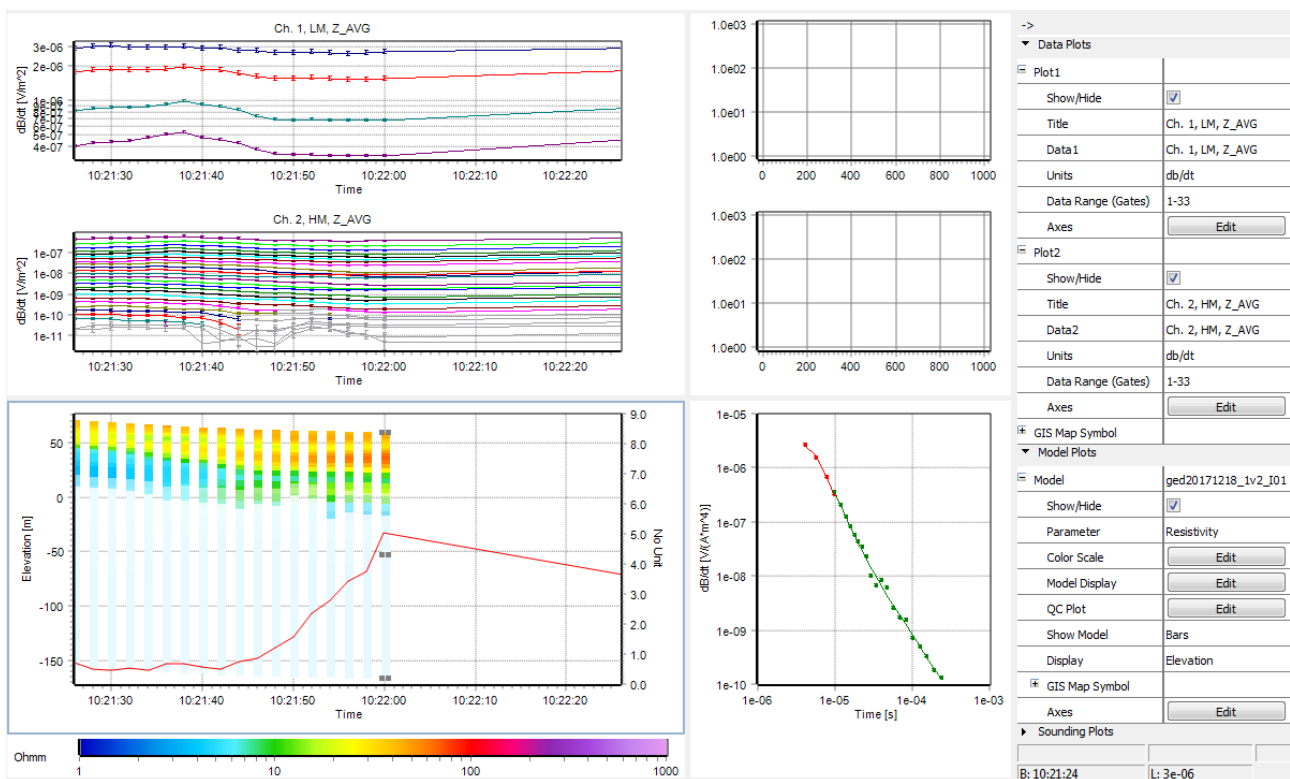

Figure 6.2: Example of some still coupled data. Ones the trouble spot has been identified, it may be necessary to adjust the splitter (grey line) between the data and model plots to get a closer view of the data.

Exercise 6.3: Evaluation of inversion results with model quality nodes

In this exercise we will take another less detailed approach to looking at the inversion result of the provisional LCI. With experience it is usually enough to only do the detailed inspection ones or maybe twice per processing node and from there use the quick overview methods given by model quality and model selection checks.

- Select the new inversion node in the workspace manager and click on **Model Quality** under the **Visualization** ribbon.
- Create model quality themes with No. of data points (for all), data residual (for all), altitude, altitude difference, DOI Standard and DOI Conservative (both depth and elevation versions) (see figure 5.2).
- Inspect the result of each quality theme. The data residual and altitude difference ones in particular. Use **Find Nearest** to jump to the relevant spots in the processing if anything suggest a closer look might be relevant.

There is one commonly used composite model quality theme that is not made with a model quality node. It is sometime referred to as a model selection check.

- Select the new inversion node in the workspace manager and click on **New** (model selections) under the **Inversion** ribbon.
- Plot both the processing node and the model selection. Have the model selection on top. Use Edit Display to ensure they have different colors. What does this show us?
- This is a very useful map when combined with **Find Nearest** to check the processing along roads and power lines.

Figure 6.3: The model quality setup. Use the F1 wiki help to read more about the setup.

Exercise 6.4: Evaluation of inversion results with mean resistivity maps
In this exercise we will take one last approach to looking at the inversion result. This one is mainly used with large SCI when all other methods have been done a few times.

- Make resistivity maps to look for non-geological structures.
- Select the model selection, go to the **Visualization** ribbon and choose **Theme**.
- Create some horizontal mean resistivity maps (**Mean Parameters**) with relevant elevations. Use the **Histogram** button to set a good upper elevation. Do a few select maps with an interval length of 10 m to cover most of the models. Blind at DOI standard. Press **Apply** and **Close**. This is not the final maps so a few select maps will be enough.
- For the final maps, one would of course make a more complete set of maps and include depth maps as well. Note that they don't have to be made with one command, it is possible to make some in 5 m intervals and then some deeper ones in 10 m intervals and so on.
- Select the model selection and press **2D Grid**. Select all the elevation themes we made earlier. Use the search filter and select all.
- Use the following settings:
    - Search radius 400 m. Large enough to reach nearby lines.
    - Node spacing 25 m. No need to make this smaller than your sounding distance.
    - Use logarithmic data transform. Use inverse distance for speed or use Kriging for nicer looking maps.
    - Use the resistivity.avc color scale.
- Now the batch process will start and make all the grids. Press Close when done.
- Click on some of the new map layers and check for non-geological structures.
- Why is this only done after all the other options?

- Some prefer to use point themes instead of mean resistivity maps. Select one of the already made themes. Then press **Point Theme** on the **Visualization** ribbon. Select the relevant images, give them a color scale and an appropriate Size. Then press OK.
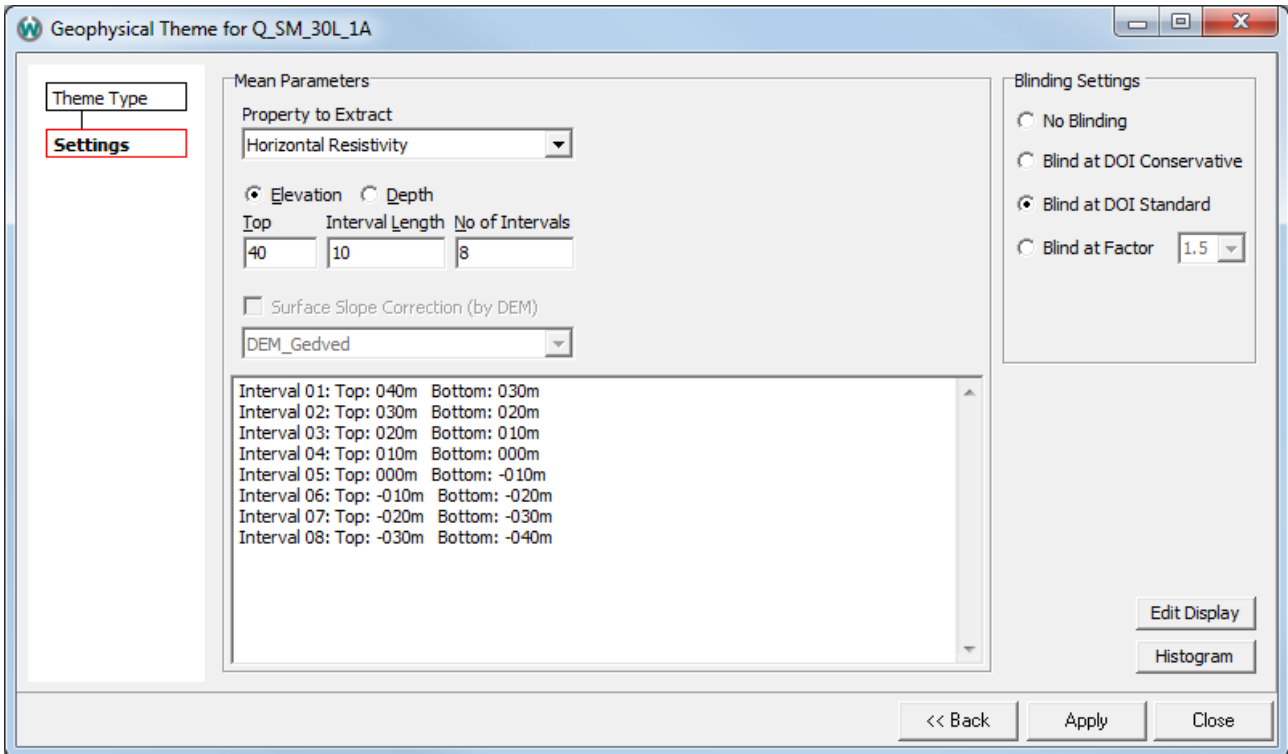
Figure 6.4a: Creating mean resistivity themes. Use the F1 wiki help to read more about the setup.
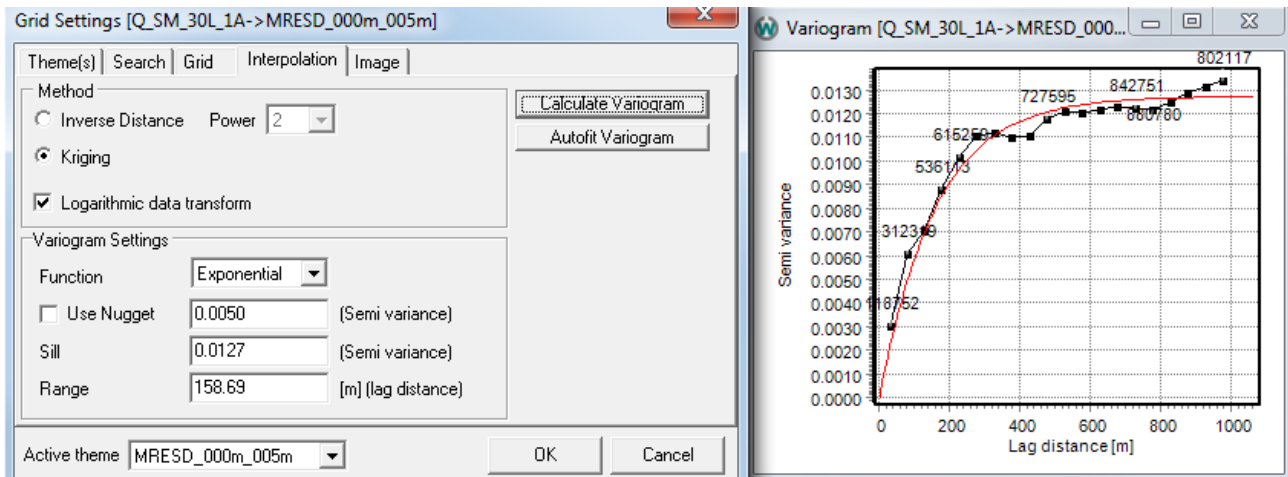

Figure 6.4b: 2D Grid setup. The variograms will be fitted individually and automatically. Use the F1 wiki help to read more about the setup.

Extra Exercise – Making your own color scales

In this exercise we want to make a new color scale.

- Select a mean resistivity grid and press **Image** on the visualization ribbon. This time press **Edit** to edit the color scale. This opens the color scale wizard. Remember that it is possible to press F1 to access the wiki help if needed.

- The first page is the design page. It holds the setup of the colors to be used, the order of those colors and a smooth or discrete option. If you for instance simply want a resistivity color scale with the reverse colors, that is easy to make.
- The second page is the map data page. It connects the raw colors to the map data. For themes there is a preview of the image with the connections. Often one will want to edit the **user defined data max/min** to an appropriate range and save the color scale. But it is also possible to change how the colors will be distributed in greater details.

Exercise 7. SCI Inversion

In this exercise we will setup a small inversion of the dataset we have processed in the previous exercises. This time as a spatially constrained inversion (SCI). Keep in mind that SCI only should be used after doing provisional LCI. Many of the options are identical between SCI and LCI, it is mainly the order in which they are set that is different.

Exercise 7.1: SCI setup
In this exercise we will setup a small smooth SCI.

- Click on **SCI** on the **Inversion** ribbon.
- Set the altitude options and check the channels (usually just z component) that should be inverted under **Channels and Gates**. Set the minimum number of data points.
- Use **Data within region** to select a small SCI area on the map where you have processed the data fully.
- Use a smooth model. Try to edit the default model to have 20 layers with a first layer boundary in 3 meters and a last layer boundary in 300 meters. Use auto resistivity or set a uniform starting resistivity. Set the reference distance to your sounding distance. Set the inversion to use as many CPU's as you can spare. Try not checking **Run inversion when done**. Hit **Finish.**
- Since we didn't set it to run right away it will just create the SCI nodes. To run the nodes later select the SCI inversion node and click invert data on the **Inversion** ribbon.
- This launches the inversion through ABI in the background. So you can still use the Workbench although maybe a bit slower than before.

Exercise 7.2: SCI a-priori example
In this exercise we will try using a-priori from GIS as an example of using a-priori for SCI.

- Select the outer SCI node of the SCI that was setup before. Then click **New** (Inversion) on the **Spatially Constrained** ribbon. This creates an additional inner SCI node (SCI inversion node) that we now can apply the a-priori to.
- Tick on the outer SCI node in the workspace manager. Make sure that it is the last layer clicked. Now use the **Polygon** selection tool from the **GIS** ribbon to draw a polygon around a few models creating a selection of models marked on the map. Add a few more models to the selection. Usually this kind of thing is done for all models over water or something like that. Now click on **Use Selection** also on the **GIS** ribbon and pick **add a-priori to** the new SCI inversion node we just created.
- Now those models will be loaded and can be edited. Try giving all the layers a new resistivity with a-priori 99, so that it only affects the starting resistivity.
- Note that it is possible to save the selection from here as well as the edited model.

- If you already have a saved selection, this kind of a-priori can be added by selecting the SCI inversion node and then clicking **GIS** on the **Spatially Constrained** ribbon.
- The other a-priori options are also used by selecting the SCI inversion node and then clicking on the relevant option on the **Spatially Constrained** ribbon.

Exercise 8: Sections

In this exercise we will visualize our inversion results on sections and see how such options can be used for comparison with other data types.

A quick node on terminology. For the most part we talk about sections, but profile also used when we talk about the actual line of the map with the location of the section.

Exercise 8.1: Simple section

In this exercise we will take a first look at the use of sections. We will begin slowly to better understand all the parts.

- Hide the mean resistivity map, or change the layer order, to see the sounding positions. Then use **Draw** on the **Sections** ribbon to draw a profile along one of the lines with data.
- This is just a line on the map, it adds a profile node in the workspace explorer, but showing by checking the box in front of it only opens an empty section.
- Select the profile from the workspace explorer and then use (Add Layers) **Models** on the **Section** ribbon.
- Here we can add models to the section (see figure 8.1). It can be done directly from inversion nodes or from the earlier made model selection. This include inversions and model selections made from entirely different data types. Use the resistivity.avc color scale and a fitting search radius. The search distance has to be big enough to cover the distance from your drawn profile to the sounding positions that you want to include, but also not so big that you get positions from more distant lines. Use the Show Bars option.
- Then go into **Model Display Settings** – **Model Bars** and set it to Elevation. Also in here we have the blinding options, change the color fading to be done with DOI standard instead of DOI conservative. Then press OK to close this and again to add the models.
- Note how this added a layer to the workspace explorer under the profile that now can be used to show and hide this layer on the section. Such layers are with a few exceptions drawn from first to last. The order can be changed by moving the layers using the **Move Up** or **Move Down** buttons on the **Workspace** ribbon.
- Show the profile with the models and move the mouse along the section to see where the models are on the GIS map.
- Experiment with the section setup:
    - Adjust the axes by clicking on them or right-click on the section and pick Format and the relevant axis. Try changing the bottom axis from distance to UTMX or UTMY.
- Experiment with the model display. This can be done by adding the models to the profile in a different way, but there are also some options for changing existing layer nodes:
    - Change search distance.
    - Change the width of the bars. This is an example of something that can be done to an existing node. Select the layer node and then click **Edit Display** on the **Workspace** ribbon to access the settings.

> o   It is also possible to change the color scale using the option in the bottom of the section itself.

Rather than drawing the profiles by hand we often use a different approach.

- Try selecting an inversion node and then use (Create New) **Models** from the **Sections** ribbon. This option makes it possible to create profiles that follow specific lines in the inversion result, directly from the model positions. With clever use of the profile direction and groups of lines, we can usually get all the relevant profiles made very quickly. Try making a few profiles.
- Note how it jumps directly to adding models and how search distance here isn't needed to the same degree as the profile will be right on top of the models.

It is also possible to add profiles based on coordinates or external grids.
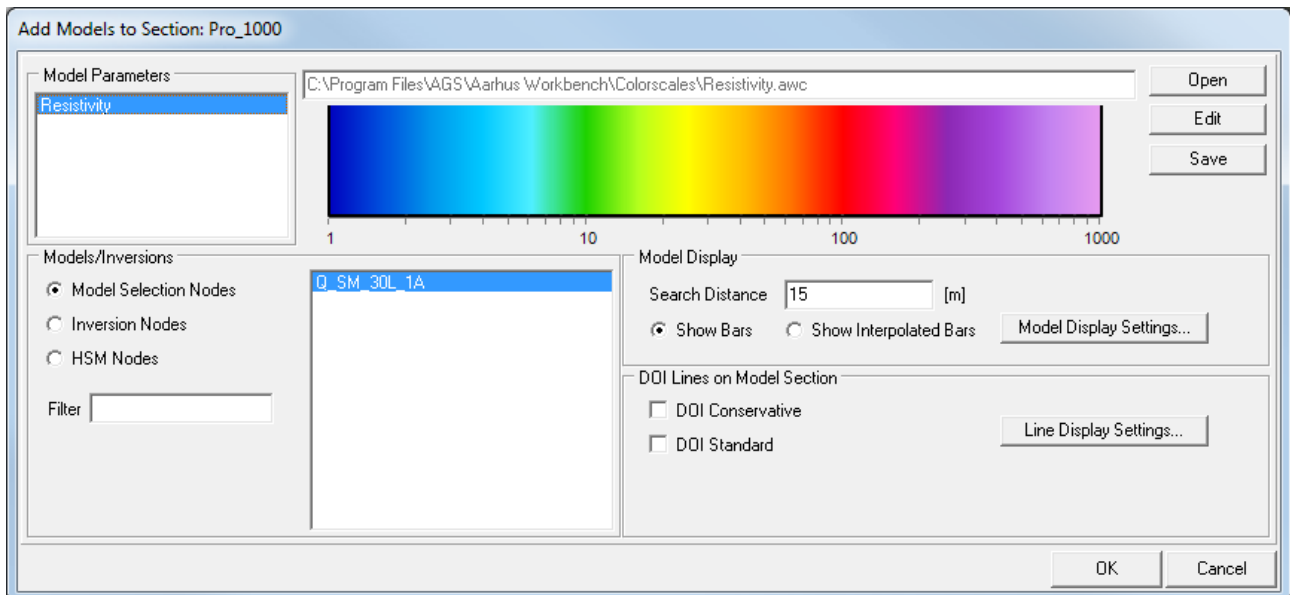

Figure 8.1: Adding Models. Use the F1 wiki help to read more about the setup.

Exercise 8.2: More about section layers

In this exercise we try to add a few other things to our sections.

- In addition to models it can be interesting to add other things to the section for comparison. This is often in the form of borehole with lithology information. The details for how to format such information and import it can be found on the wiki. Here we will assume that such a data-base exist and we just need to connect to it.
- Go to database explorer (it is below the tab for the workspace explorer). Click on Lithological data. Then click **Open Database** on the **Database ribbon**. The select the provided database with lithological data. Now click the **New** (Borehole Selection) button on th**e Database** ribbon.

- A window will give options to filter which boreholes to include. Boreholes can be added within a certain area which can be selected by dragging a square on the GIS map or by setting a minimum borehole depth. All boreholes will be imported if none of these settings are enabled. Import all boreholes. When done, this will add a node in the workspace explorer.
- Select the Borehole Selection in the Workspace Explorer and press **Edit Display** on the **Workspace** ribbon to change the size and color of the GIS points to different from model positions.
- Try drawing a relevant profile and add some models to the section. Then again select the profile and use **Lithological Logs** on the Section ribbon to also add the borehole.
- The same can be done with **Geophysical Logs** if you have any of those, except that this is done under Geophysical data not Lithological data.

- Another thing often added is grids. Any surfer grid can be added to the workspace as **External Grid** (use **Elevation Model** if is it topography) and then added to the sections with **Grids as Line** on the **Sections** ribbon. We sometimes also use this option to show the data residual of the models on the sections.
- Select the data residual model quality node made earlier. Then use **2D grid** just as was done earlier only this for sure use inverse distance. We don't necessarily care about the image in this case. Ones we have the grid node, select the profile and add the grid with **Grids as Line** on the **Sections** ribbon.
- It is also possible to add an **Image**, most commonly used with seismic lines. And to add either **Interpolates 3D Grids** or actual **3D Grids**.

Exercise 8.3: Explore the copy options for Sections using a master section

For larger surveys it is often practical to setup one section and then apply the same setup to the other sections.

- When drawing a new section, it is possible to point to an existing master and get the new section created with the same layers. Try it.
- Select a section and press (Copy) **Section** on the sections ribbon to make a copy of a section so that you can have two versions of the same section open at ones. This is useful when comparing different data types on the same section.
- Select a section and press **Layers** on the sections ribbon to copy the layers of that section to the selected section.
- When working with a very large number of Sections, it is often useful to select the main profiles node and press **Batch** (Add Layers) on the sections ribbon to do the copying. Experiment. Try for example to adjust the left axis of your master section from auto to a nice looking fixed interval, and have that applied to the other sections you have made.