



User Manual: TEMImage Beta software

Beta Version 4.0.3.6

TABLE OF CONTENTS

1. Introduction.....	2
2. Data Import	4
3. Data Processing	5
4. Data Inversion.....	7
5. Viewing Inversion Results	8
6. Saving and Customizing the Model Plot.....	11
7. Adding a new online background map	12

1. INTRODUCTION

This document explains how to import, process, and invert time-domain data collected using the **sTEM**, **sTEM Profiler**, and **TEM2Go** instruments. TEMImage is optimized for data collected in a line and is particularly well-suited for generating profile sections.

Before You Begin

Please review the following requirements before proceeding:

- TEMImage is compatible **only** with data collected using TEMcompany products. Other data formats are not supported.
- It requires datasets recorded in **dual-moment mode**; both Low Moment (LM) and High Moment (HM) data must be included.
- For best performance, use the **original project or data folder structure** as downloaded directly from the instrument.
- A valid **software license** is required for the program to run.
- To request a license, please contact: support@temcompany.com.

Once TEMImage is installed and the license is uploaded, the **startup screen** will appear as shown in Figure 1.



Figure 1: Startup screen displayed after launching TEMImage

Menu Bar Overview

The menu bar, located in the upper left corner, provides access to the following options (Figure 2 and Figure 3):

- **File** – Import, export, save, and access settings.
- **Visualization** – Opens a new window for manually creating profiles and resistivity layer maps, after inversion has been performed (see Section 5).
- **Options** – Includes advanced processing selections and the option to update DEM (elevation data).
- **Help** – Displays application version details and license status.

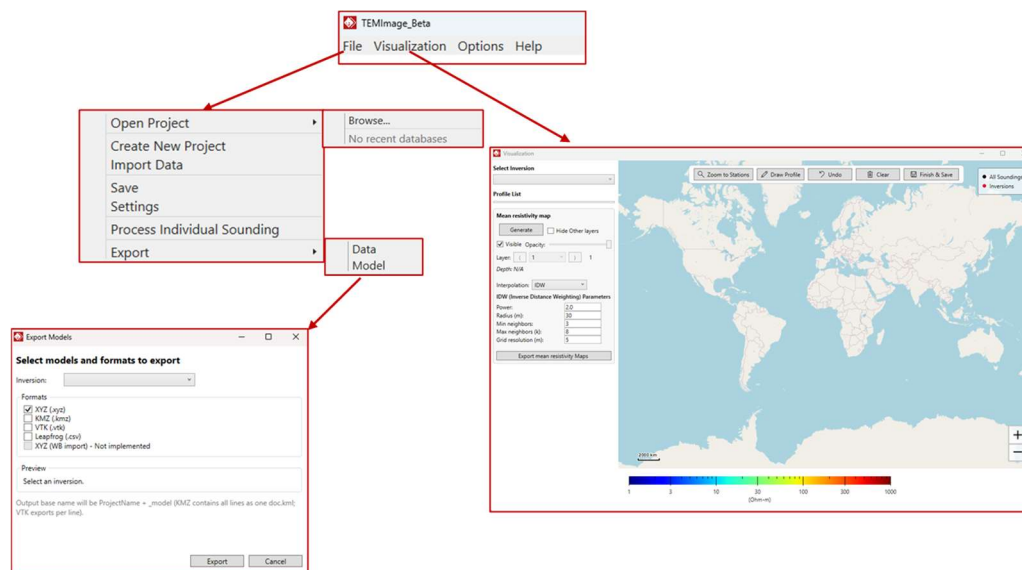


Figure 2: Overview of File and Visualization menus

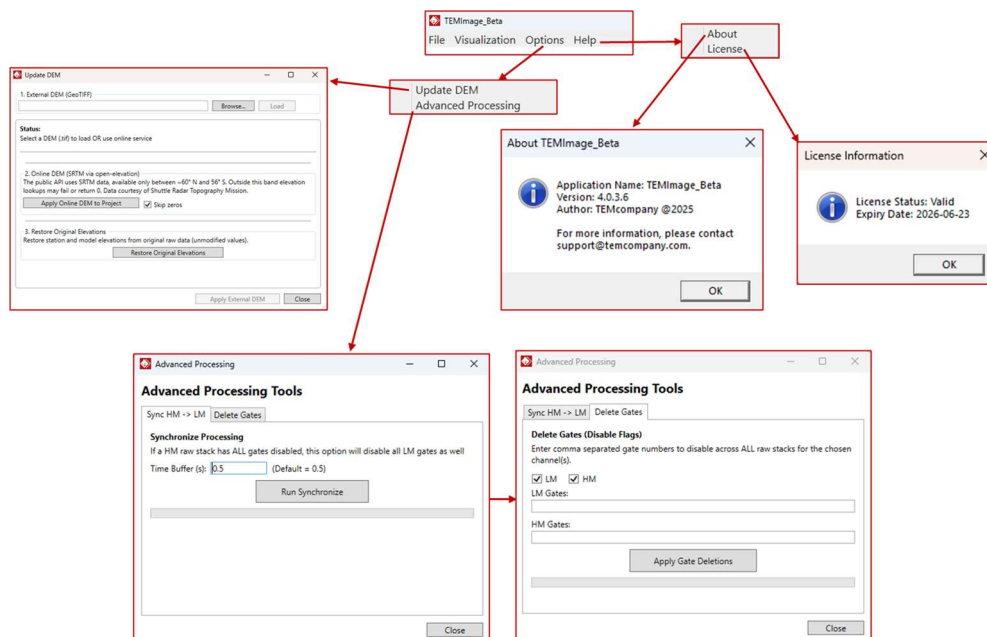


Figure 3: Overview of Options and Help menus

2. DATA IMPORT

To begin, click on **File** and select **Create New Project** (Figure 2). A window will open prompting you to create a database file (.db), where all processed data will be stored.

Once the database file is created, an **Import Data** window will appear (Figure 4). Here, you must:

- Select the data type (*sTEM/sTEM Profiler*, *TEM2Go* or *tTEM*).
- Choose the **Data** folder containing the acquired measurements.

Importing as sTEM or sTEM Profiler

When importing sTEM or sTEMprofiler data, TEMImage automatically performs the following:

Reads all .stb and .stn files from the selected Data folder(s)

Groups data by line number. If a dataset is missing line number information, it is assigned to the default line number **9999**. Such datasets can later be merged or reassigned after import.

Processes and stacks the raw data into soundings, including associated GPS positions. TEMImage does not support datasets without GPS information.

Importing as TEM2Go (or tTEM)

When importing TEM2Go data, additional options become available

- Select Production Line File (.lin)
If no .lin is selected, then TEMImage will split the data automatically.
- Processing Options – Station Interval (m)
Choose the spacing you want the raw measurements averaged into

After import, both **Low Moment (LM)** and **High Moment (HM)** data displayed, along with a map showing the location of your acquired measurements (Figure 4). All GPS data allocations are shown on the map in black. The currently highlighted line is displayed in red. This setting can be modified by **right-clicking on the map** and using the **Map Settings** option.

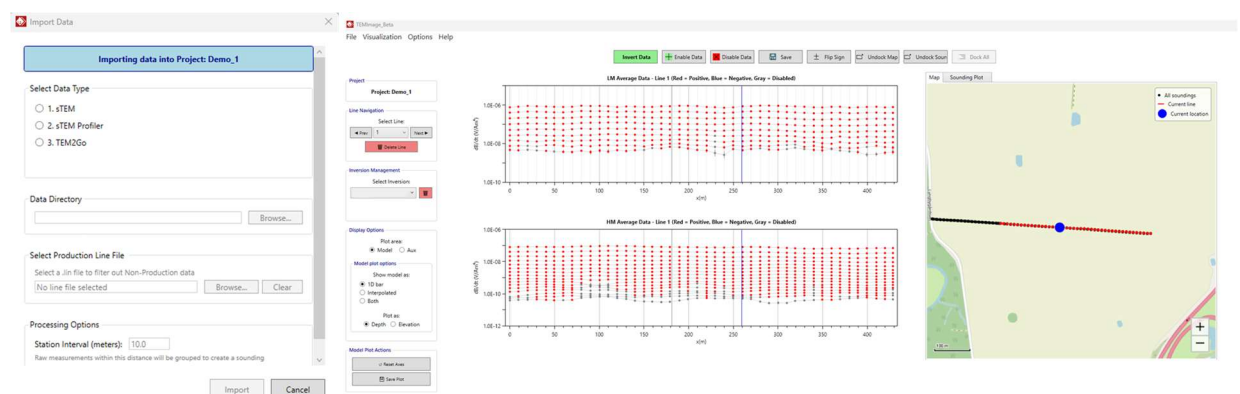


Figure 4: Import data window (left) and overview of the home screen after importing (in this case) TEM2Go data (right).

3. DATA PROCESSING

When data is imported, TEMImage automatically applies some default filters. After this step, you can manually review and adjust the data by enabling or disabling specific data points in the **Low Moment (LM)** and **High Moment (HM)** profile data plots (Figure 5). This helps ensure that only good-quality data is used for the inversion. In projects containing multiple survey lines, you can switch between them using the **Line Navigation** option in the panel on the left (Figure 5).

You can manage data points in two main ways:

1. Using the Data Profiles

- Left-click and drag the mouse to select the data points you want to modify on the profile. The selected points will be highlighted (Figure 5).
- From the top menu, choose **Enable** (or **Ctrl+E**) to include the points in the inversion process, or **Disable** (or **Ctrl+D**) to exclude them.

2. Using the Sounding Plot

- When you select a data point on a profile, its corresponding sounding is displayed in the **Sounding** tab (Figure 5).
- Click the Sounding tab to view the details. Data can be displayed in either **dB/dt (LM and HM)** mode or **Apparent Resistivity** mode.
- You can also select data points directly in the sounding plot and enable or disable them in the same way.

Notes:

- Pay close attention to data points near roads, houses, or other infrastructure, as they are more likely to be affected by noise.
- Disabled data points are excluded from the inversion, they are never deleted.

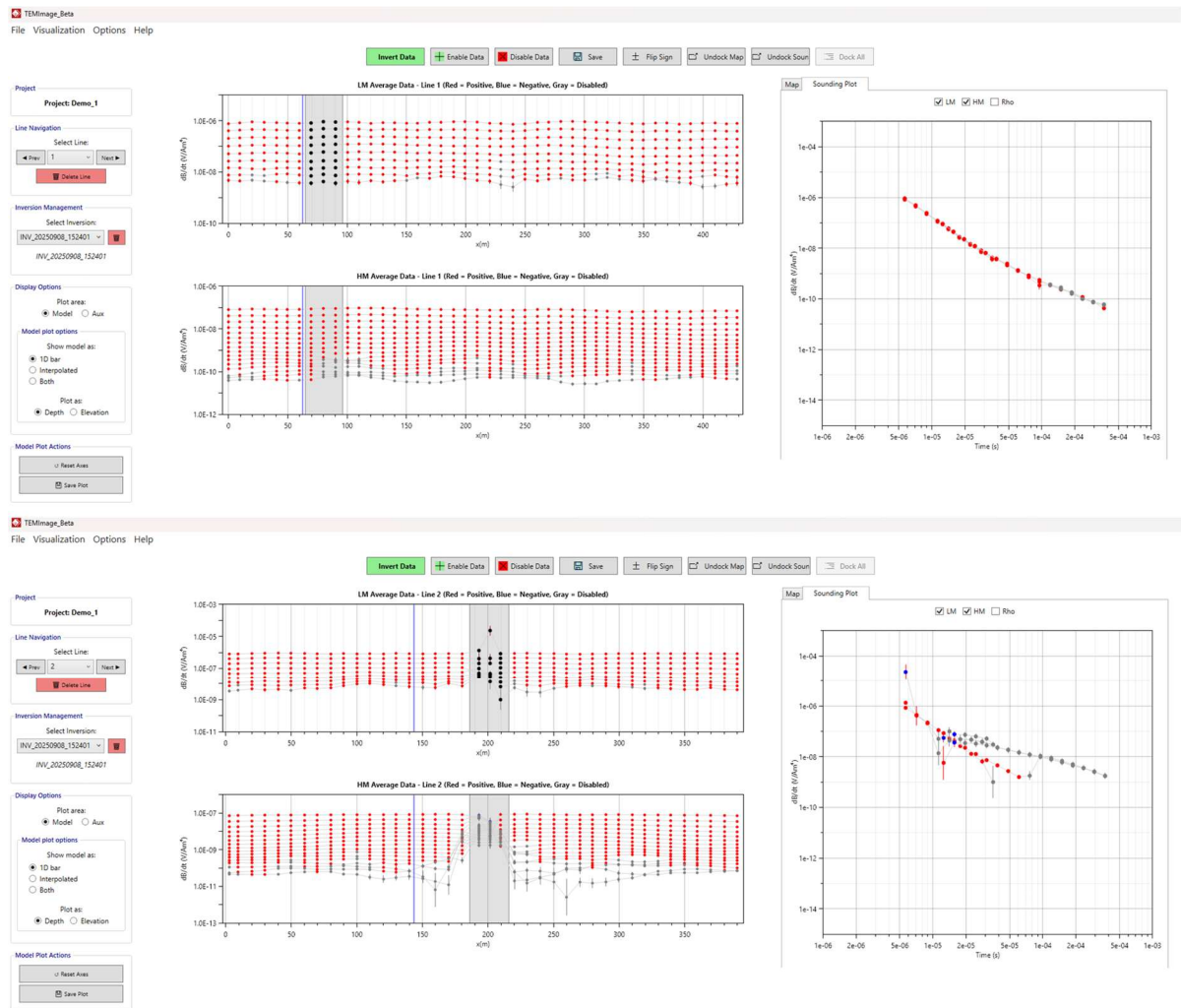


Figure 5: Representation of TEM2Go data acquired along a survey line. Data points are selected from the profile plots on the left, while the corresponding soundings are displayed on the right in the Sounding tab (shown here in dB/dt). *Line 1 (top)* illustrates an example of good-quality data, whereas *Line 2 (bottom)* shows data affected by coupling, which should be disabled before running the inversion.

4. DATA INVERSION

Data inversion is performed using the **Lateral Constraint Inversion (LCI)** scheme together with a smooth layer model.

When you have completed the data processing, click on the **Invert Data** button in the top panel (Figure 5). This will open the Inversion Setup window (Figure 6), where you can:

- Select one or multiple lines to invert.
- Adjust inversion settings, including the number of layers, first- and last-layer boundaries, constraint type, and inversion type.

Once the inversion begins, a small progress window will appear (Figure 6) showing:

- Current and previous iteration time
- Current iteration and misfit

The inversion process will automatically stop once the stop criterion is met. When complete, the resulting model is displayed in the bottom panel (Figure 7).

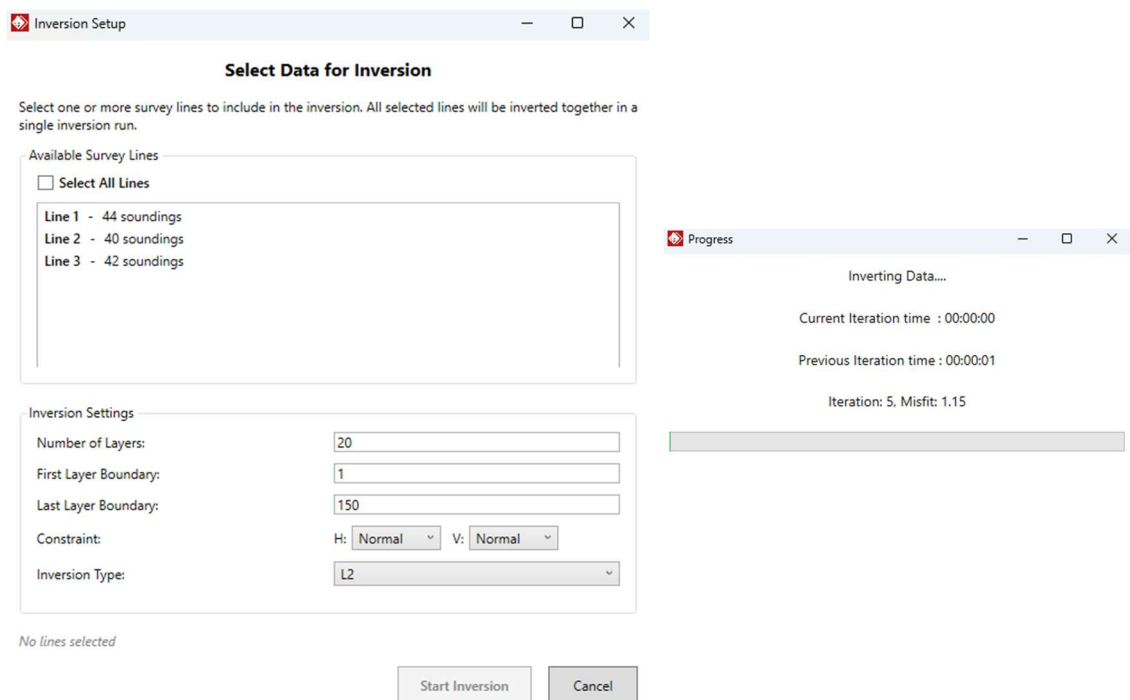


Figure 6: Inversion Setup window (left) before starting the inversion and Progress window (right) after the inversion process has started

5. VIEWING INVERSION RESULTS

The resistivity section created as a result of the inversion (Figure 7) can be displayed in three different ways (Figure 8):

- **1D Color Bars**
- **Interpolated Section**
- **Combined View** (1D color bars overlaid on an interpolated section)

If your project contains multiple soundings or survey lines, you can switch between resistivity sections using the **Line Navigation** option.

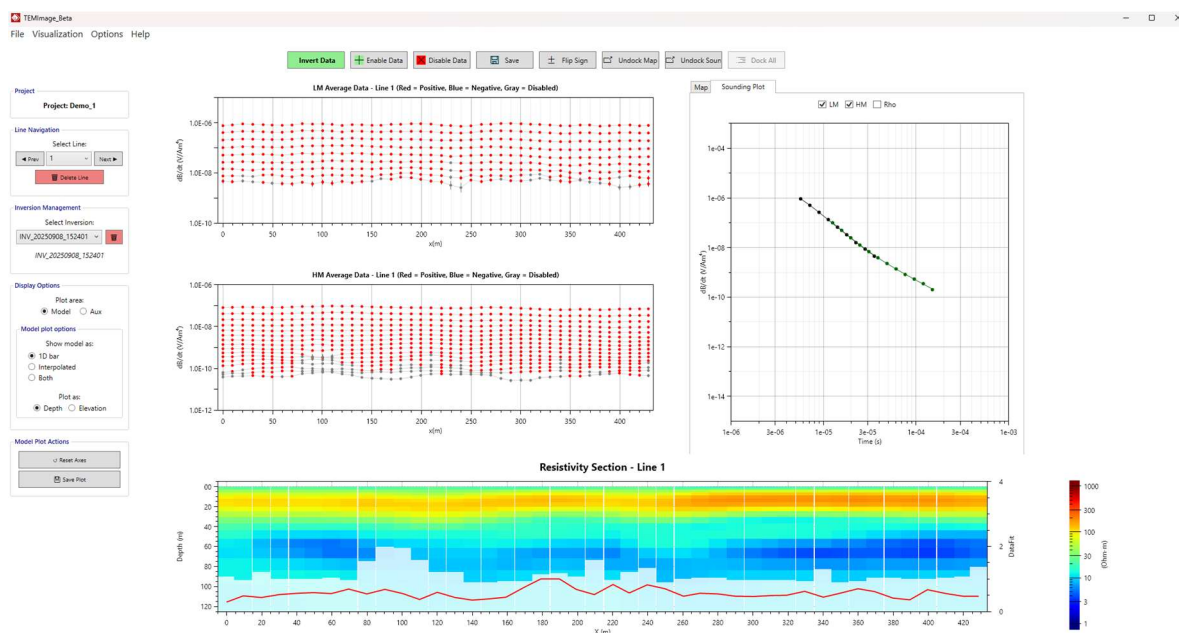


Figure 7: Home screen after the inversion of the data is complete. The bottom panel shows the Resistivity section for the corresponding line. If you click on a sounding on the Resistivity section, the Sounding plot on the right will show how well the forward model fits the observed data.

Other useful information that can be obtained from the resistivity section, includes:

- **Depth of Investigation** - model layers below the estimated depth of investigation are faded in white (Figure 7) to indicate their reduced sensitivity to the measured data.
- **Data Misfit Line** - a data misfit line (red line) is plotted on the y-axis of each section model (Figure 7).
 - A consistently high misfit value (e.g., 1.5 or above) suggests the presence of poor-quality data that should be reviewed and possibly disabled.
 - To check the misfit for a specific sounding, left-click on the model. The Sounding plot will display how well the forward model fits the observed data (Figure 7).

If inversion results are unsatisfactory, you can further process the data, disable problematic points, and re-run the inversion.

Managing Multiple Inversion Models

Each time you run an inversion, a **new inversion model section** is created for the selected line. When the inversion is complete, TEMImage automatically switches to the latest model.

To review earlier results, use the **Inversion Management** panel (located below Line Navigation on the left) to select and view previous inversion versions.

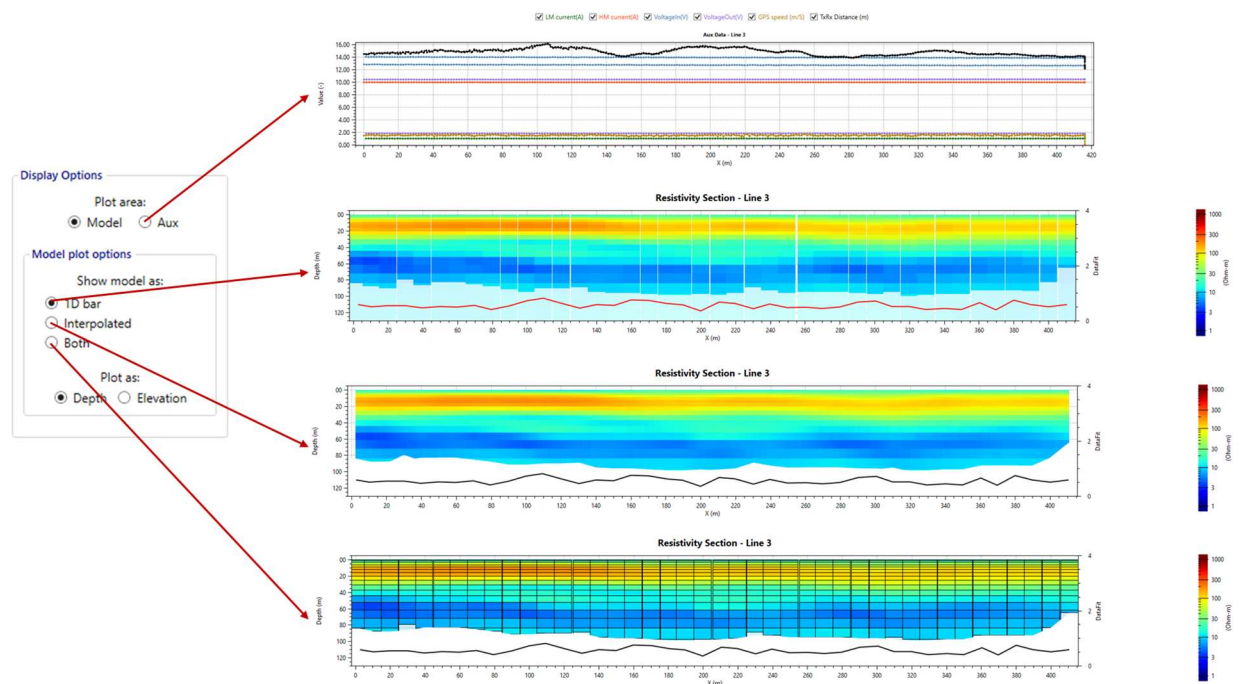


Figure 8: Display Options panel (left). Selecting different options changes what is shown in the bottom panel of the Home screen. The top illustration represents the *Auxiliary Plot*, which provides information about LM/HM current, input/output voltage, GPS speed, and Tx–Rx distance during data acquisition. The other illustrations show the three available display modes for the model plot: 1D Soundings, Interpolated Section, and Both (combined view).

Creating custom profiles

TEMImage allows you to draw and generate custom profiles directly within the survey area.

To create a profile (Figure 9):

1. Open the **Visualization** menu. A new window with a map of the acquired data will appear.
2. Click the **Draw Profile** button in the top panel.
3. On the map, click to set your profile's starting point, then click again to set the endpoint. A line will automatically be drawn between the two points.
4. When satisfied with the profile, click **Finish and Save** in the top panel.

5. Enter a profile name and specify a search radius in the pop-up window, then click **OK**.
6. The profile will appear in the **Profile List** on the left panel of the Visualization window.
7. To open your profile, double-click it. A **Profile Plot** window will appear, where you can adjust display settings.

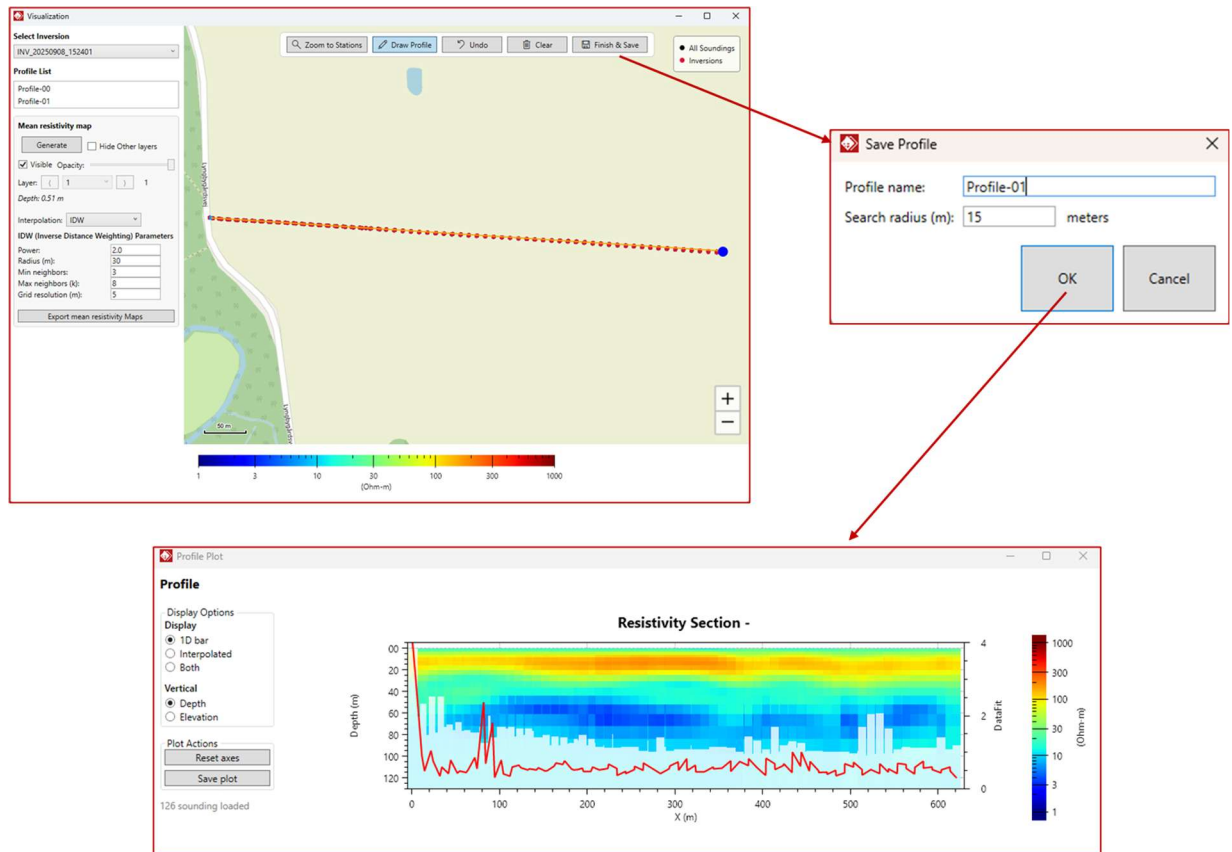


Figure 9: Visualization window and creating custom profiles.

Generating Mean Resistivity Maps

When working with dense survey areas, you can generate **Mean Resistivity Maps** to visualize depth slices across the project area. The maps are created by interlating the resistivity values of a particular layer at regular grid interval defined in Inverse distance (IDW) interpolation parameters.

To generate a map (Figure 9):

1. In the left panel of the Visualization window, click the **Generate** button under *Mean Resistivity Map*. It will generate the maps using default IDW settings.
2. TEMImage will create a layer view of resistivity slices at different depths. The depth shown is the vertically centred depth between top and bottom of that layer.
3. Use the same panel to adjust settings such as opacity, interpolation method, and slice depth.

6. SAVING AND CUSTOMIZING THE MODEL PLOT

Saving the Project

Once your data is imported and processed, go to **File > Save** to store your work (Figure 2). This will save the current state of your work into the project database file.

- The next time you open the project, all the imported data and inversions will be there.

Customizing Views

You can adjust how your results are displayed by customizing the Model Plot and Color Scale.

- To do this, right-click on either element and select the settings you want to modify (Figure 10).

Saving the Model Plot

To save a model plot as an image:

1. Right-click on the model plot.
2. Select Save Plot.
3. The model will be saved as a PNG file.

Export Options

To export a model:

1. Go to the **File** menu and choose **Export > Model** (Figure 2).
2. In the pop-up window, select the inversion version you want to export.
3. Choose the desired format. Supported export formats are XYZ, KMZ, and VTK.

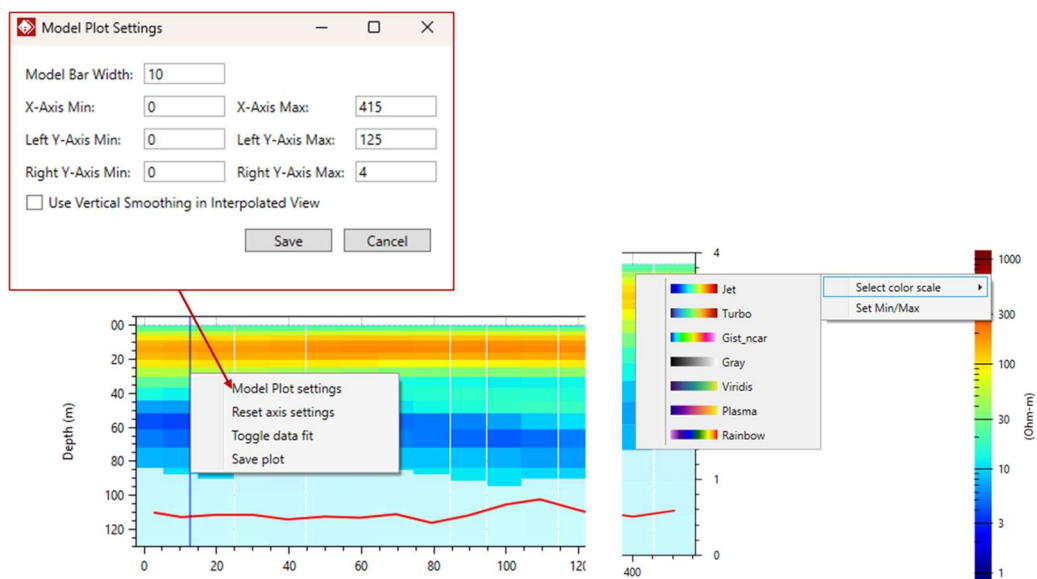


Figure 10: Customizing model plot and color scale settings

7. ADDING A NEW ONLINE BACKGROUND MAP

In this beta version of TEMImage, background maps can only be added using WMS layers. By default, the application loads the Open Street Map. However, you can add other WMS layers as long as you have a valid URL and access to it —similar to how it's done in QGIS.

Adding a new online background map (Figure 11):

1. Right-click on the map window.
2. Select "Add new map provider".
3. Enter a name for your map (e.g., "MyMap").
4. Paste the valid WMS URL.
5. If the URL is valid, the new background layer will be added automatically.

Viewing and setting the new background layer:

1. Right-click again and select "Select Map Provider".
2. You should see the newly added background layer listed.
3. Select it to set it as the active background map.

Once selected, the new map layer will be saved with your project and remain active until you choose a different one.

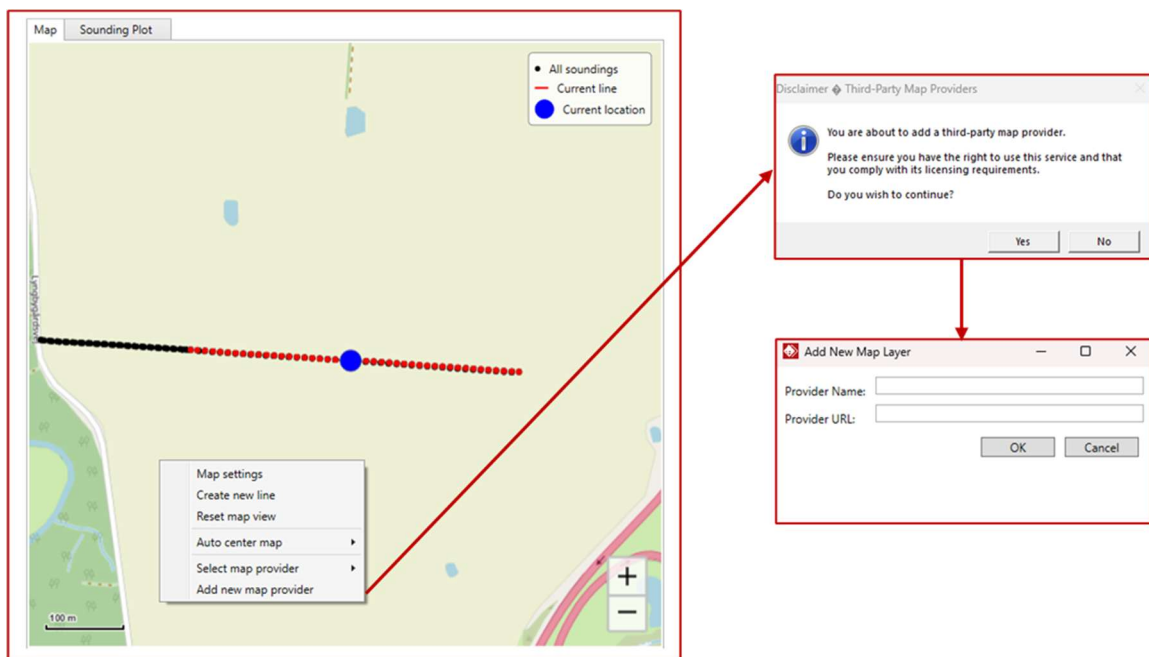


Figure 11: Adding a new background map