ERTLab Studio

Geostudi Astier - www.geostudiastier.com

Feb 16, 2024

ERTLAB STUDIO

CHAPTER

ONE

INSTALLATION

Follow the steps below to install an application.

Contact us if you have any problem.

1.1 Setup

The program can be installed with this setup:

http://www.geostudiastier.com/ViewLab3D/ViewLab3D_x64_Setup.exe

Unzip, run "*setup.exe*" and then follow the instructions.

For old 32bit computers the setup is:

http://www.geostudiastier.com/ViewLab3D/ViewLab3D_Setup.zip

In case of problem it is possible to use this alternative installer:

http://www.geostudiastier.com/ViewLab3D/ViewLab3D_Installer.exe

1.2 Prerequisites

This software can need that the following prerequisites are installed on the computer to correctly be executed.

In case of problem first try to install the following tool, then try to reinstall the software.

- Microsoft Visual C++ 2017 Redistributable Package

  https://aka.ms/vs/17/release/vc_redist.x64.exe

- Microsoft Visual C++ 2005 Service Pack 1 Redistributable Package MFC Security Update

  https://www.microsoft.com/en-us/download/details.aspx?id=26347

  – To download the correct version select "*vcredist_x86.exe*"

Please note that a final reboot can be necessary.

1.3 Dongle Drivers

If you have problems using the green USB protection dongle probably you simply need to refresh the installation of its drivers.

Figure 1.1: The protection dongle

Get the file from the following link:

https://www.keylok.com/sites/default/files/install12062022.zip

https://www.keylok.com/sites/default/files/install04092022.zip

https://www.keylok.com/support/install-utility-download

Download the zip archive, then open it, it will contain a single file named "*Install.exe*", it is the driver installer/uninstaller.

To complete correctly the procedure described remove the USB dongle from your computer. It also important to continue as Administrator, or at least to run the executable as administrator.

Note that USB 3 ports can give some problems, so if you are having difficulty, it is recommended that you try plugging the dongle into a different port (or use a USB HUB that is often USB 2).

1.3.1 Uninstall old drivers

To clean the system from bad previous installations, first it can be useful to remove already present drivers (if any). So run "*Install.exe*", a window will be displayed. Check "*Uninstall*" on the bottom of the window displayed, ignoring all the other information shown, then press the button "*Begin Uninstall*" and follow the instruction provided.

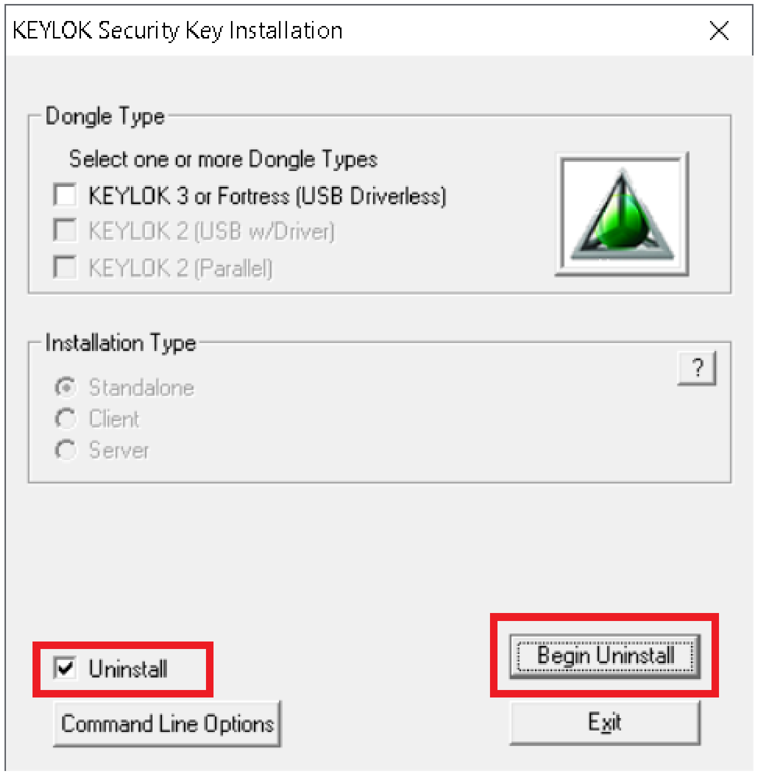


Figure 1.2: Uninstall old drivers

1.3.2 Install new drivers

When the previous step is completed, run again "*Install.exe*", and fill the fields as shown in the following picture:

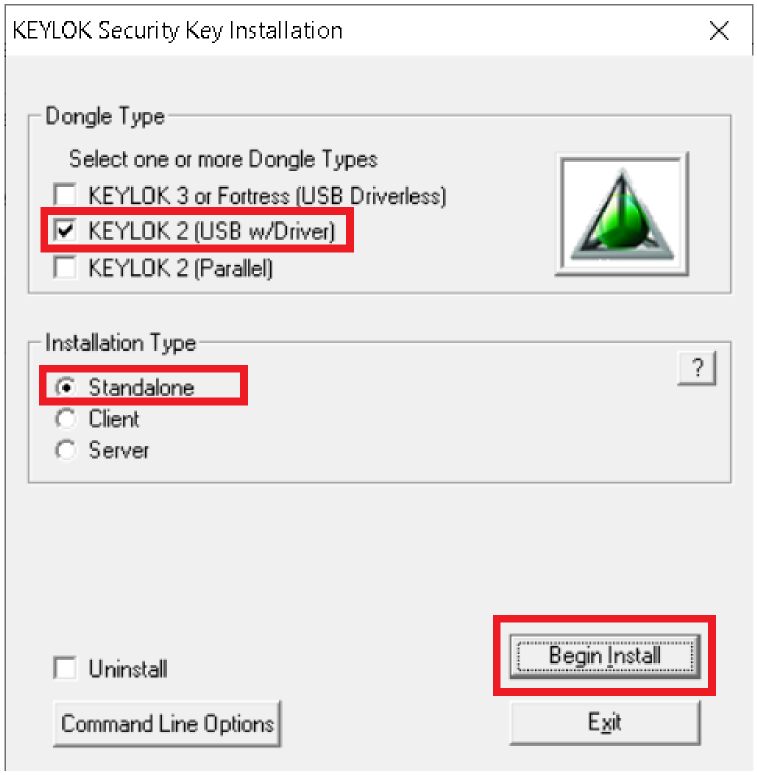


Figure 1.3: Install new drivers

The following message is displayed, so please remove the USB dongle from your computer before continue.

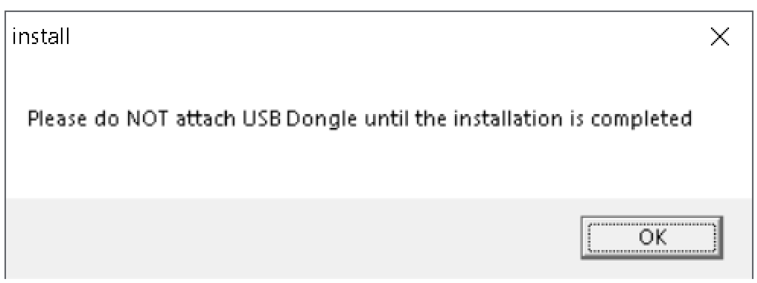


Figure 1.4: Message

Press "*Begin Install*" and follow the instructions displayed.

1.3.3 Open the software

At the end, you can insert the USB dongle into a free port, then wait 5/10 seconds, then open the software. If the procedure was done correctly any error message will be displayed.

1.4 Tutorial

1. Quick 2d ERT Profile Inversion Process
   - English:

     http://www.geostudiastier.com/ViewLab3D/Manual/ERTLabStudio_Tutorial1_ENG.pdf
   - Italian:

     http://www.geostudiastier.com/ViewLab3D/Manual/ERTLabStudio_Tutorial1_ITA.pdf
2. ERT Data processing
   - English:

     http://www.geostudiastier.com/ViewLab3D/Manual/ERTLabStudio_Tutorial2_ENG.pdf
   - Italian:

     http://www.geostudiastier.com/ViewLab3D/Manual/ERTLabStudio_Tutorial2_ITA.pdf

1.5 Video tutorial

1. Fast 2D Inversion
2. (Italian only) L'inversione della caricabilità (IP)

CHAPTER

TWO

INTRODUCTION

This document describes the main features of the software **ERTLab**, produced by Geostudi Astier s.r.l. It is a software programmed to manage geophysical data, to design, processing, visualize, and analyse geoelectric data.

With *ERTLab Studio* it is possible to:

- Design the electrodes geometry: *ERTLab Studio* manages any type of array configuration, also unconventional configuration (as loop around buildings);
- Generate 2D/3D sequences for surface, cross-borehole or surface-to-borehole electrode setups;
- Compute the forward model of resistivity and chargeability;
- Compute sensitivity analysis;
- Compute the optimized sequence for multichannel instruments;
- Import ERT data from a variety of sources;
- Compute a pre-processing of the measurements through advanced features such as statistical analysis and filtering;
- Create a custom *mesh*: it is possible to choose the dimension, the cells size, the boundary conditions, and there is the possibility to import a topography file and to insert stratigraphic layers;
- Set starting model with a lot of customization options;
- Compute data inversion;
- Compute operation between file (useful in case of time-lapse survey);
- Manages 4D projects.

2.1 Licenses

The software has different license levels:

- **Basic Level of Licence**: **ViewLab 3D** works as a viewer and integrates ERT data and models in a single 3D graphical area. In the 3D visualization mode it is possible to visualize data through the insertion of custom sections, volumes, surface, and to integrate them with images, three-dimensional graphic objects, and DXF files.
- **Advanced Level of Licence**: **ERTLab Studio**: works as a resistivity/chargeability/IP inversion software, managing every phase of an ERT survey, from the survey design to the pre-processing and inversion of collected data including forward models. All the Basic Level licence functions are included.
- **Graphic Pack**: This is an add-on to enable the following features:
  - Delimiter
  - Model Fence Section
  - Geographic Map Image
  - Core Sample
  - Point Cloud
  - Metal Factor
- **4D**: This is an add-on for **ERTLab Studio** only, to enable the Timelapse 4D Project.

CHAPTER

THREE

USER INTERFACE

3.1 Main interface

After installing **ERTLab Studio / ViewLab3D**, insert the proper dongle key in the PC and open the application; the following "splash screen" will display Figure 3.1.

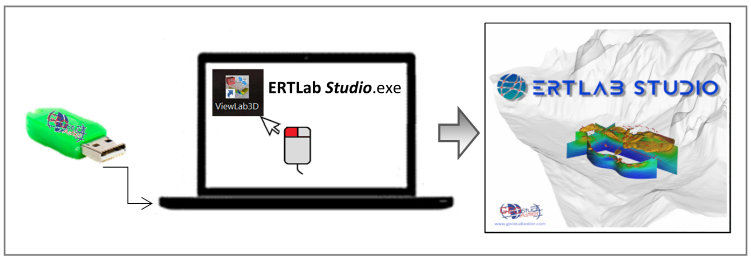


Figure 3.1: On the left, USB protection dongle; on the right starting splash screen

After a few moments, the application will open showing the main interface, which is shown in Figure 3.2.

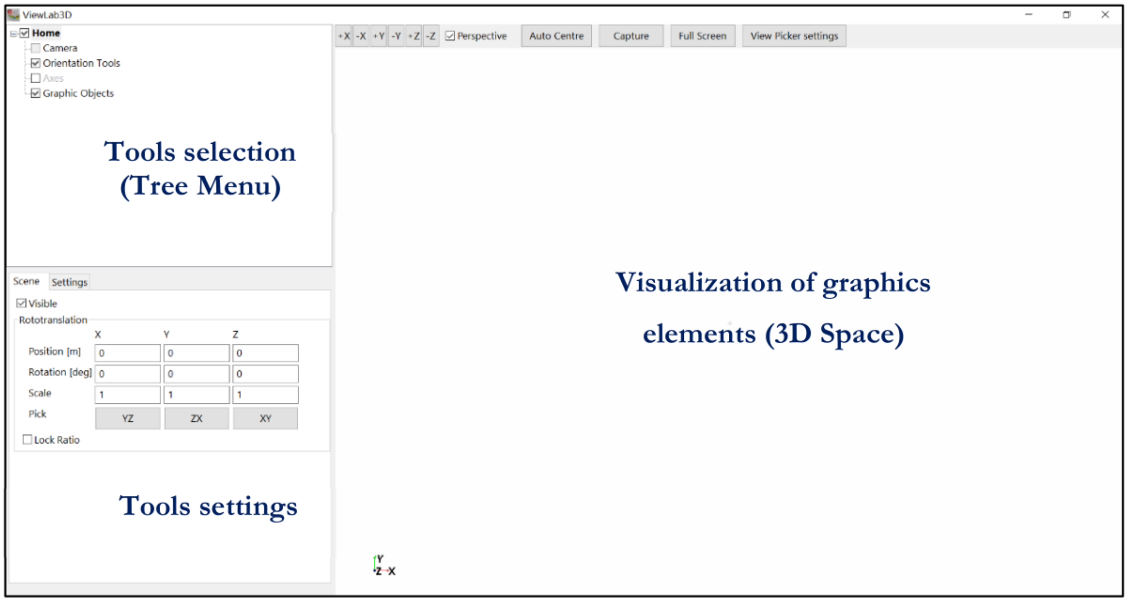


Figure 3.2: Interface at the software opening

This is divided into **3 main areas**:

- At the top left there is the **tree menu** (Figure 3.3), which allows to select the object to act on and to operate and manage an entire project. At the beginning of the work the some tools already show in the tree, but when a project will be loaded further elements will appear.

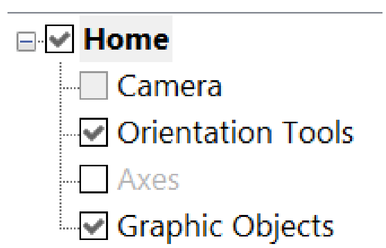


Figure 3.3: Tree menu at the beginning of the work

- At the bottom left of the screen (Figure 3.3), various **tools** will appear when one submenu of the tree is selected. Through these tools it is possible to manage the properties of the various graphic elements shown on the right. Therefore, the contents of this part of the screen changes depending on the selected tree menu and the license level installed.
- On the right (Figure 3.3) is the **3D space** where the elements selected in the tree menu are shown. If no project or object is loaded, it is empty. To easily understand how to interact with the scene in the next paragraph an object has been placed into it. The creation, the insert mode, their settings and other items will be discussed later.

3.2 Interaction with 3D area

The panel of the 3D scene is interactive; it is possible to move the shown objects with the mouse. It is possible to carry out different actions by clicking in different areas of the panel, as illustrated in the following figure (Figure 3.4).

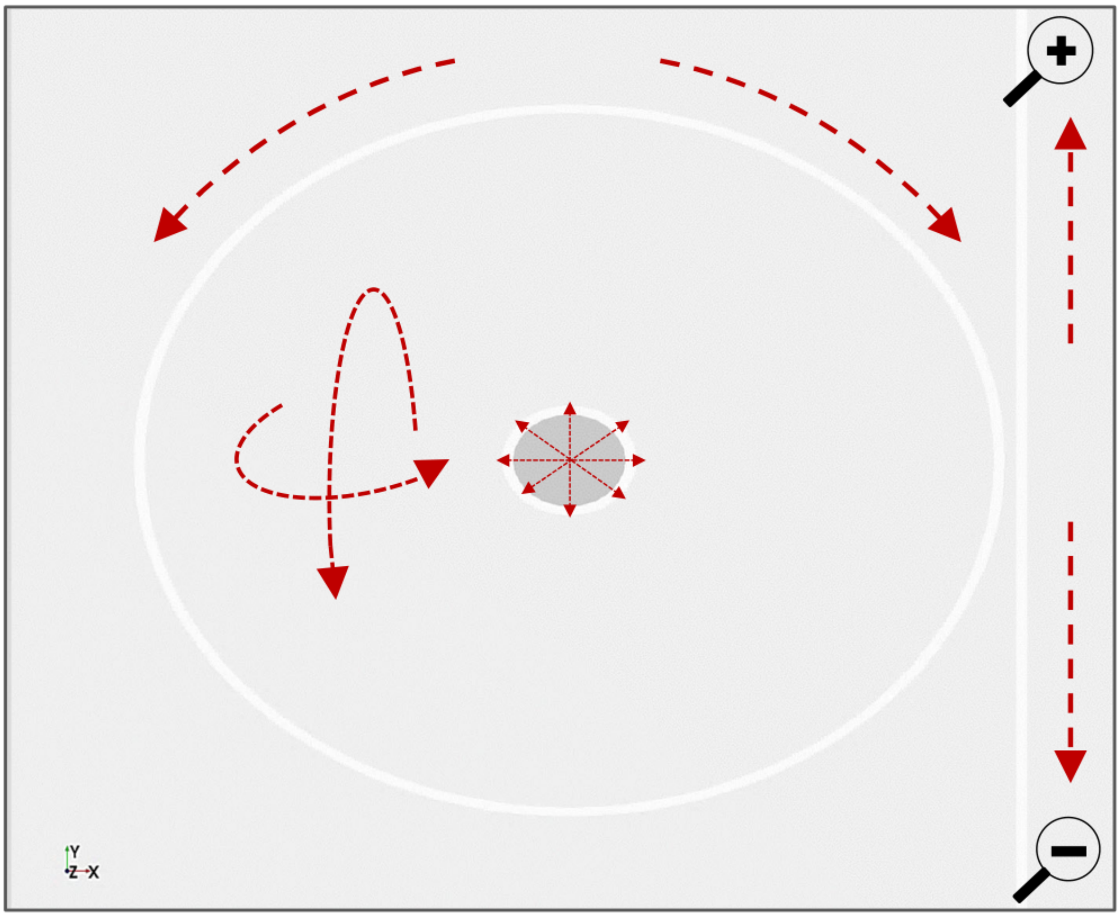


Figure 3.4: 3D empty scene; each area allows a specific movement of the object, as it is suggested by the arrows

The area is divided into four main parts; each of them with a specific function:

3.3 Translation

Clicking at the centre of the scene, a little circle of light grey will appear; clicking inside this inner circle and holding the left mouse button the object on the scene will rigidly shift as one unit, as shown in Figure 3.5. The dashed arrow suggests the direction for dragging the mouse). To easily understand how to interact with the scene, an object (House) has been placed in it. How to create an object with the insert mode will be discussed later.

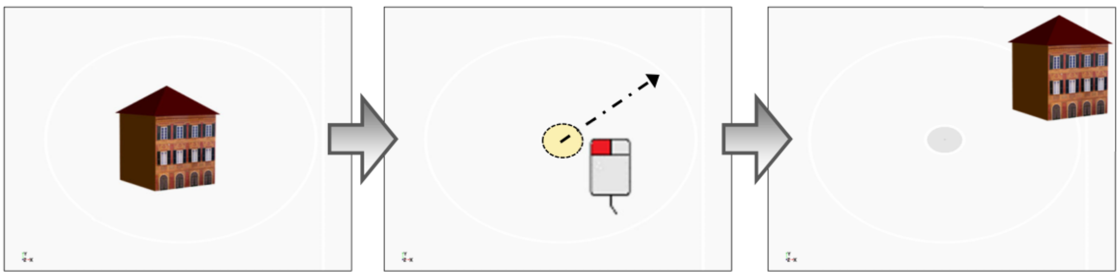


Figure 3.5: Diagonal shift of the object originally positioned at the centre of the scene, acting on the yellow area of the panel

This is the procedure to move an object even if it is not located at the centre of the scene, and the inner circle is not overlapping it (Figure 3.6).

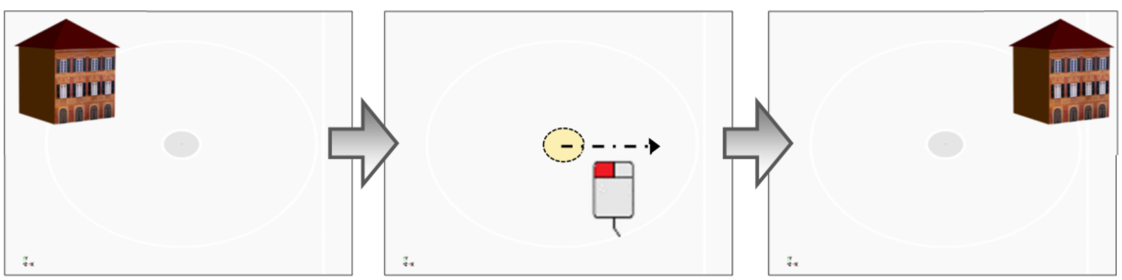


Figure 3.6: horizontal shift of the object originally positioned at the top left of the scene, acting on the yellow area of the panel

3.4 Rotation

Outside the little inner circle there is another bigger one. Through it the 3D scene can be rotated horizontally, dragging the mouse to the left or to the right, or vertically, moving it upwards and downwards. It is possible to start the movement in any points within the big circle highlighted in yellow in the following . To rotate the object, hold the left mouse button and drag the mouse inside this area; when the button is released the rotation stops.

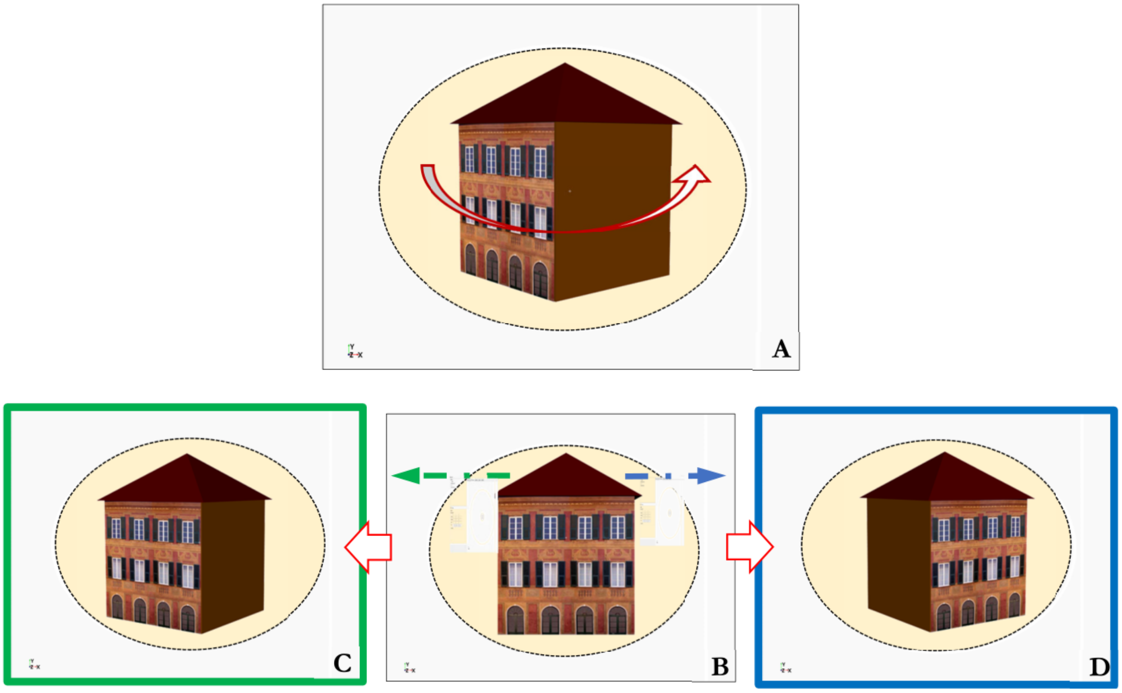


Figure 3.7: In A an object (House) is in the scene in a casual position; the rotation direction is suggested by the arrow. In B the house in original position; in C and D rotation of the house in anticlockwise and clockwise horizontal directions

In an example shows the horizontal rotation. Starting from a frontal point of view of the House (Figure 3.7 B) and dragging to the left the mouse while holding the left button, the house rotates anticlockwise (Figure 3.7 C, in green). On the contrary, by dragging the mouse to the right the house will rotate clockwise (Figure 3.7 D, in blue). The rotation direction is suggested by the arrows in Figure 3.7 A.

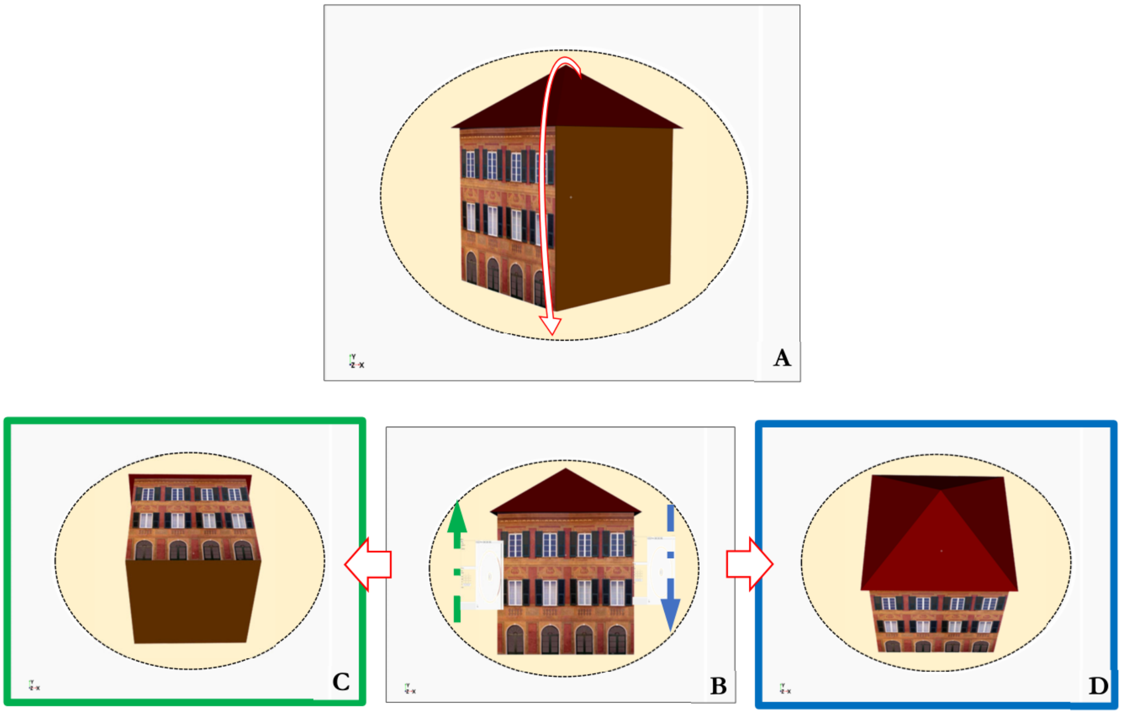


Figure 3.8: In A an object (House) in the scene in a casual position; the rotation direction is suggested by the arrow. In B the house in original position; in C and D rotation of the house upwards and downwards in vertical directions

The same applies to vertical rotations (Figure 3.8). Starting from a generic point of view of the House (Figure 3.8 B) and dragging the mouse upward while holding the left button, the house rotates vertically upward (Figure 3.8 C, in green). On the contrary, by dragging the mouse downwards the house will rotate downwards (Figure 3.8 D, in blue). The rotation direction is suggested by the arrows in Figure 3.8 A.

3.5 Spin

The outer part of the panel rotates the object right and left, in the direction parallel to the panel, as shown in Figure 3.9. Starting from a frontal point of view our example house (Figure 3.9 B) and dragging to the left the mouse holding the left button, the house will be rotate anticlockwise (Figure 3.9 C, in green). On the contrary, by dragging the mouse to the right the house will be rotates clockwise (Figure 3.9 D, in blue). The rotation direction is suggested by the arrows in Figure 3.9 A.

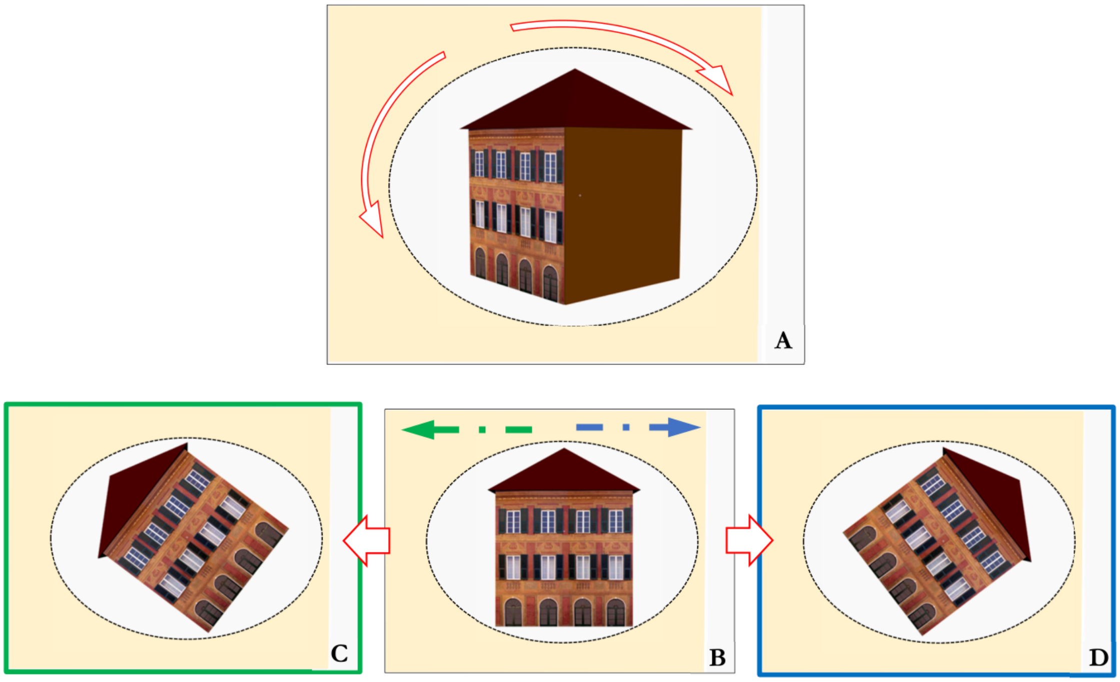


Figure 3.9: In A the object in a casual position with the direction of rotation suggested by the arrow; in B house in original position, in C and D rotation of the house in anticlockwise and clockwise direction, parallel to the yellow panel

3.6 Zoom

At the right side of the panel there is a vertical band which lets you zoom in (shifting upwards) and zoom out (shifting downwards) the object in the scene. To do this, drag the mouse vertically on the band, holding the left mouse button and releasing it when the desired zoom level is reached; it is also possible to achieve the same results through the use of the mouse wheel in each point of the 3D area (Figure 3.10).

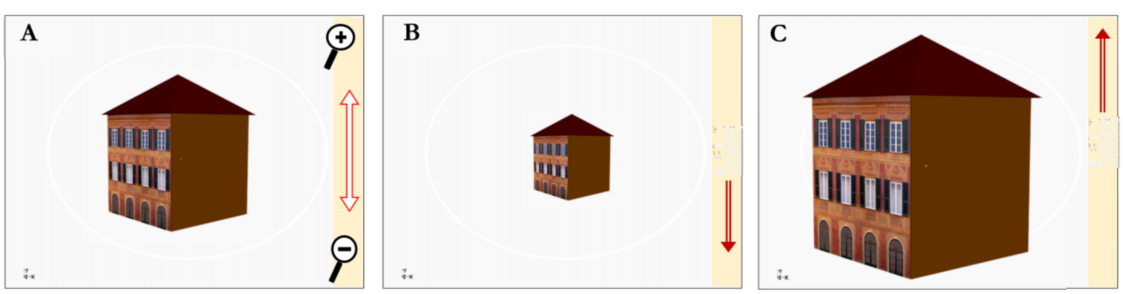


Figure 3.10: In A the object in the original position, with the direction of rotation suggested by the arrow; in B zoom out of the house, in C zoom in of the house

CHAPTER

FOUR

TREE MENU

This is the main menu, in a tree structure. It allows to manage each element of the 3D scene when one submenu in the tree is selected. Some dedicated functions will appear on the panel at the bottom of the screen. **Home** is the main menu of the tree. When no project is loaded yet, it is composed by four secondary menus: *Camera*, *Orientation Tools*, *Axes*, and *Graphic Objects*. The *ERT Data* menus will appear when a project is loaded or created. These menus expose further submenus with a lot of dedicated tools. This manual will be described them in detail. To easily understand how the properties described below modify the scene, a cube has been placed created (Figure 4.1). Again, the inserting mode of items and their setting will be discussed later.

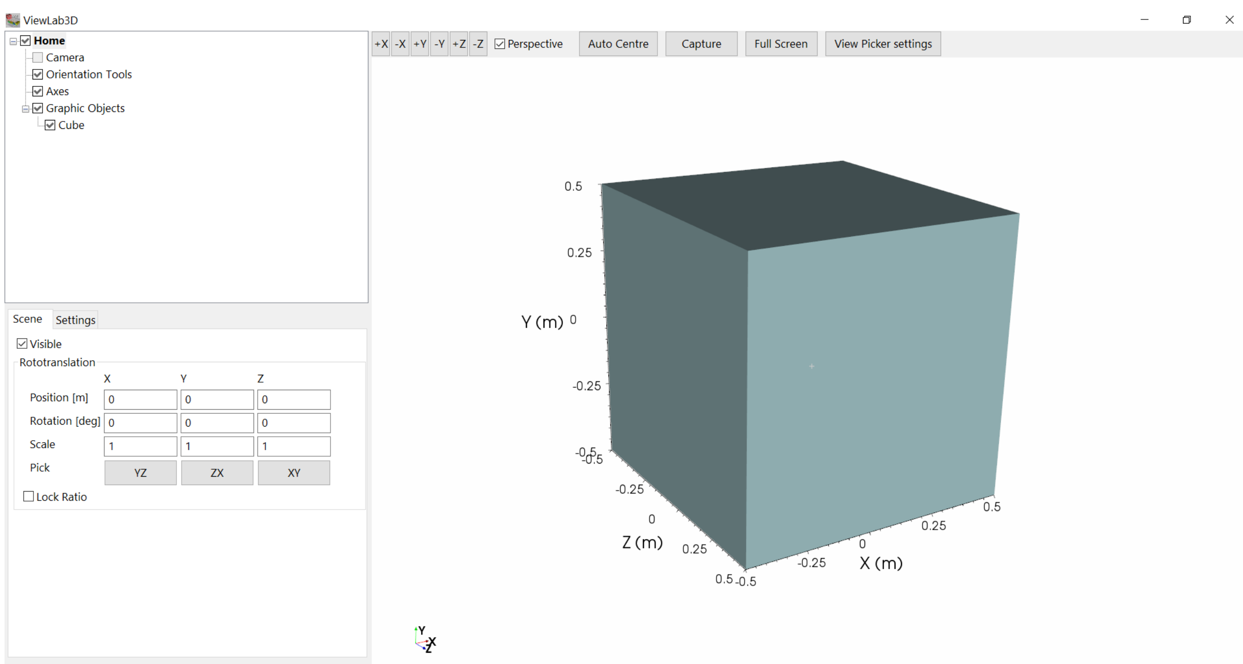


Figure 4.1: A cube in the scene

Clicking on *Home* node, two panels will appear at the bottom of the screen, *Scene* and *Settings*:

4.1 Scene

It manages the *Rototranslation* parameters of the entire 3D project. The related panel is shown in Figure 4.2.

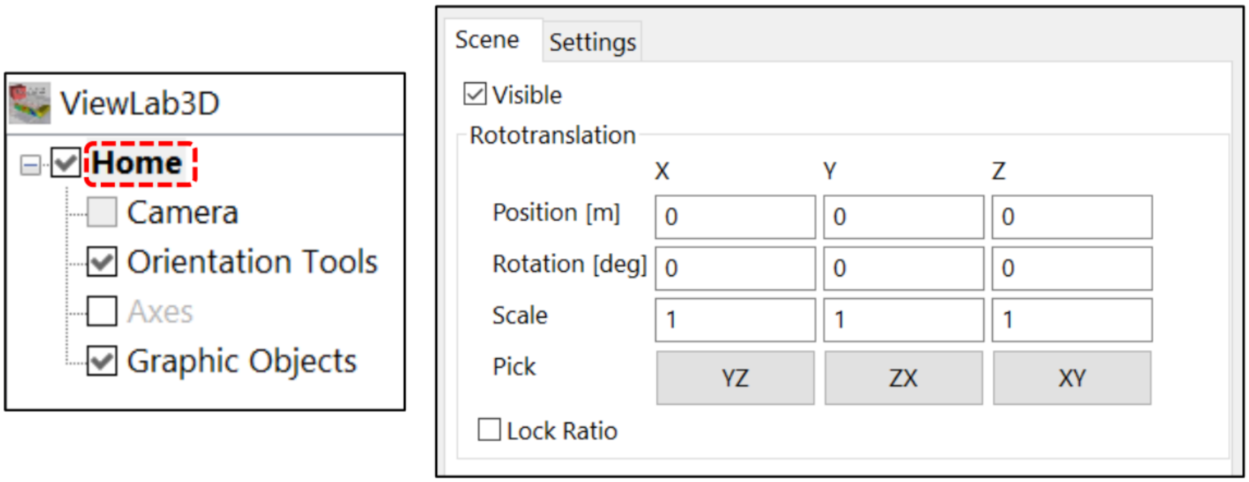


Figure 4.2: Scene Panel

It has the following items:

- *Visible*: It makes the whole scene visible if checked otherwise not.
- *Rototranslation Table*: To apply to all the objects a transformation operation (see Rototranslation)

4.2 Settings

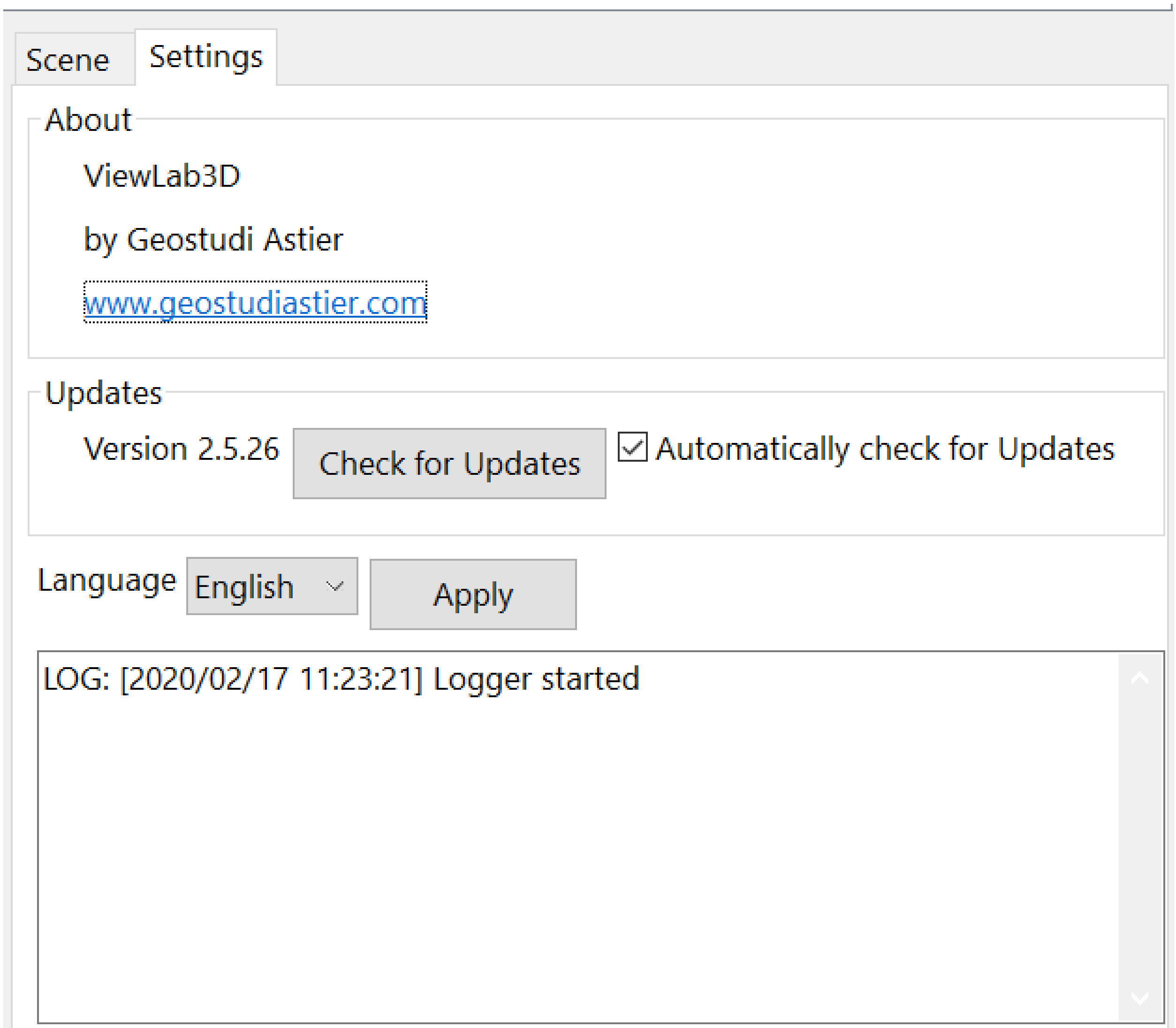


Figure 4.3: Setting Panel

The *Setting* panel (Figure 4.3) contains general information about the software, the version currently installed, a button to check for updates. If "*Automatically check for Updates*" is selected, **ERTLab Studio / ViewLab3D** will verify the availability of any updates every time the software is started. The *Languages* currently available are English, Italian, French, Spanish, German and Russian. In the lower part of the panel a text box shows the various Log information. The same information are available as text files in the subfolder "Log", which is located in the installation path.

The **Home** menu is composed of 5 submenus: *Camera*, *Orientation Tools*, *Axes*, *Graphic Objects*, and *ERT Data* project, if loaded. Each submenu has some dedicated tool and further items will be added if the *Advanced Licence* is active.

CHAPTER FIVE

CAMERA

This menu item (Figure 5.1) allows to operate on the 3D scene and to modify the visualization mode through some tools with the following functionality

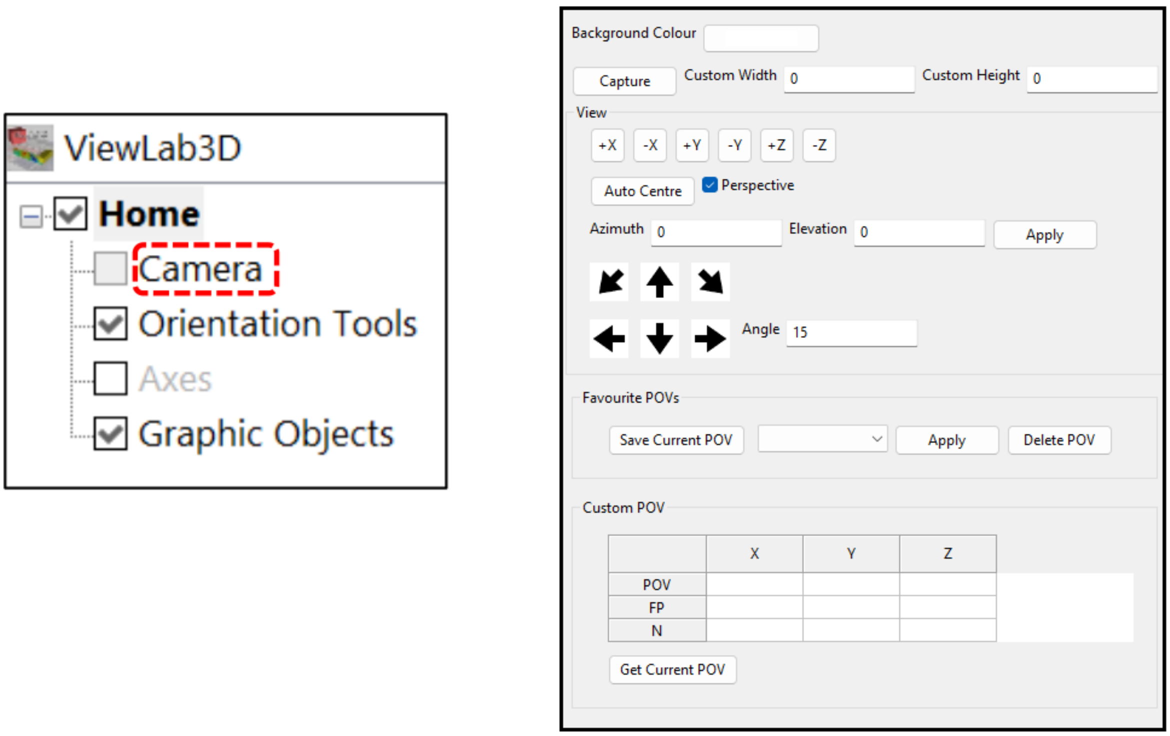


Figure 5.1: Camera panel

5.1 Background Colour

The background colour can be selected, which is white by default. As an example, in Figure 5.2 it is set to a grey and an orange background colour.

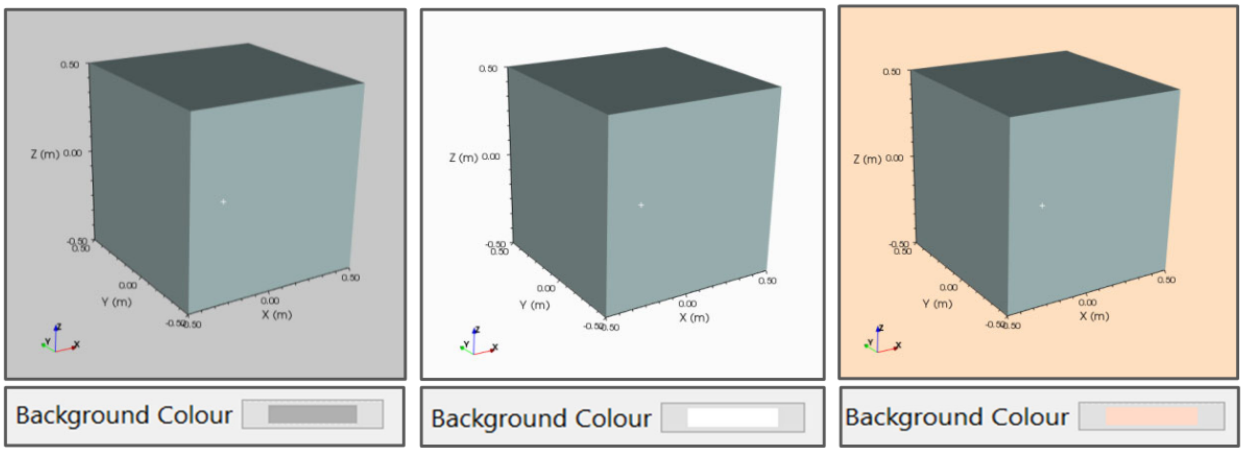


Figure 5.2: Different choices of background colour

5.2 Capture

Allows to save what is currently shown in the 3D scene as an image file; the same button is in the 3D scene, on the top of the 3D panel (Figure 5.3).

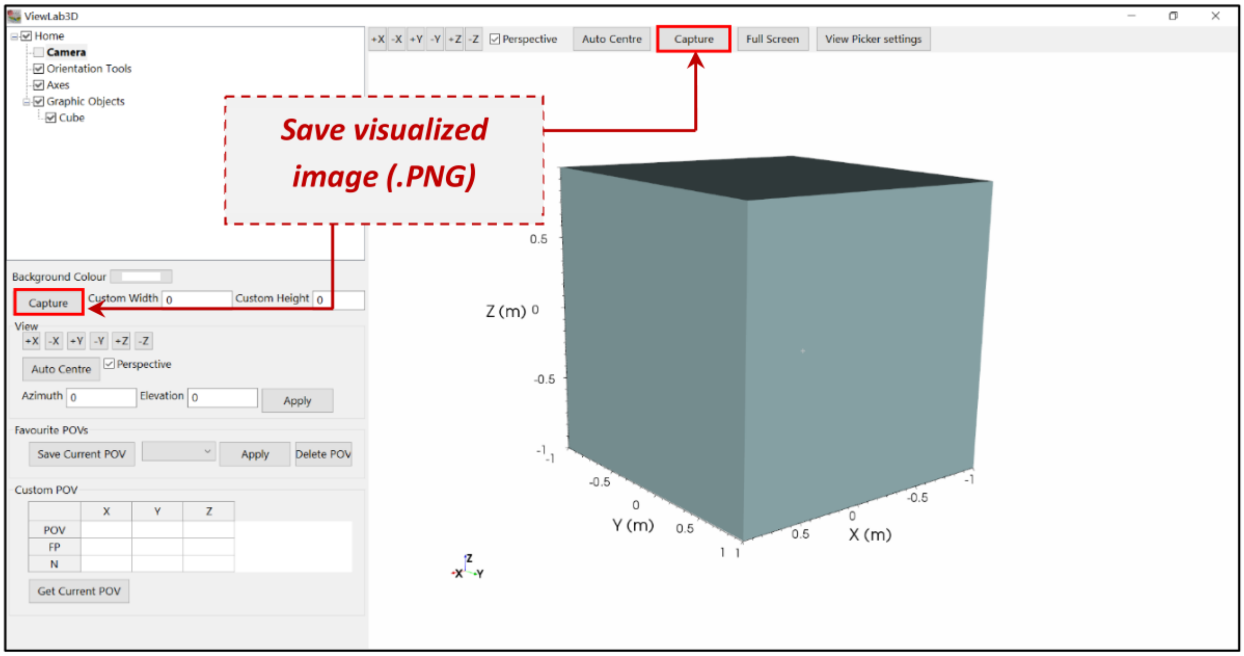


Figure 5.3: Capture tool to save the current image

It is possible to save the scene in most common raster image formats like PNG, JPEG, TIFF, BMP.

If the view is set from the top and the prospective is disabled, then it will be saved in the same folder of the image file a world file (see for example Wikipedia), that is a location reference of the image and can be used from GIS software to correctly locate the image (see for example GeoTIFF file format).

It is also available some experimental exports to 3D file format (see VRML and X3D). You can check our free viewer to share these files with your clients.

http://www.geostudiastier.com/ViewLab3D/OnlineViewer

It is possible to choose the width and the height (in pixels) of the image to save. Figure 5.4 shows what is obtained inserting, e.g. 1000 pixels as width value and 2000 pixels as height value, and vice versa.

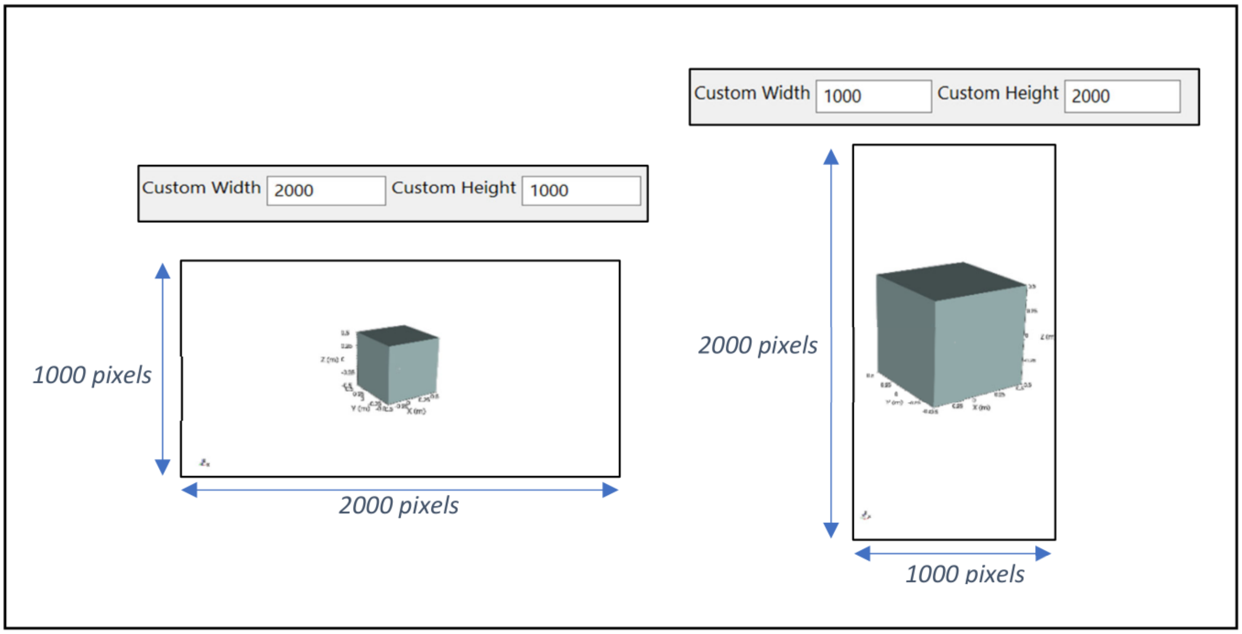


Figure 5.4: Different choices of width and height values

Please note that common image resolution can be like:

	4:3	**16:9**
VGA 360p	480 x 360	640 x 360
SD 480p	640 x 480	854 x 480
HDready 720p	960 x 720	1280 x 720
FullHD 1080p	1440 x 1080	1920 x 1080
QuadHD / 2K 1440p		2560 x 1440
UltraHD / 4K 2160p		3860 x 2160
8K		7680 x 4320
16K		15360 x 8640

5.3 XYZ buttons

Allow the user to rotate the scene along one of the three axes (X, Y, Z) and to specify from which direction ("-" and "+") to observe it. As an example, three objects (a cube, a sphere and a pyramid) are observed from different points of view as shown in Figure 5.5.

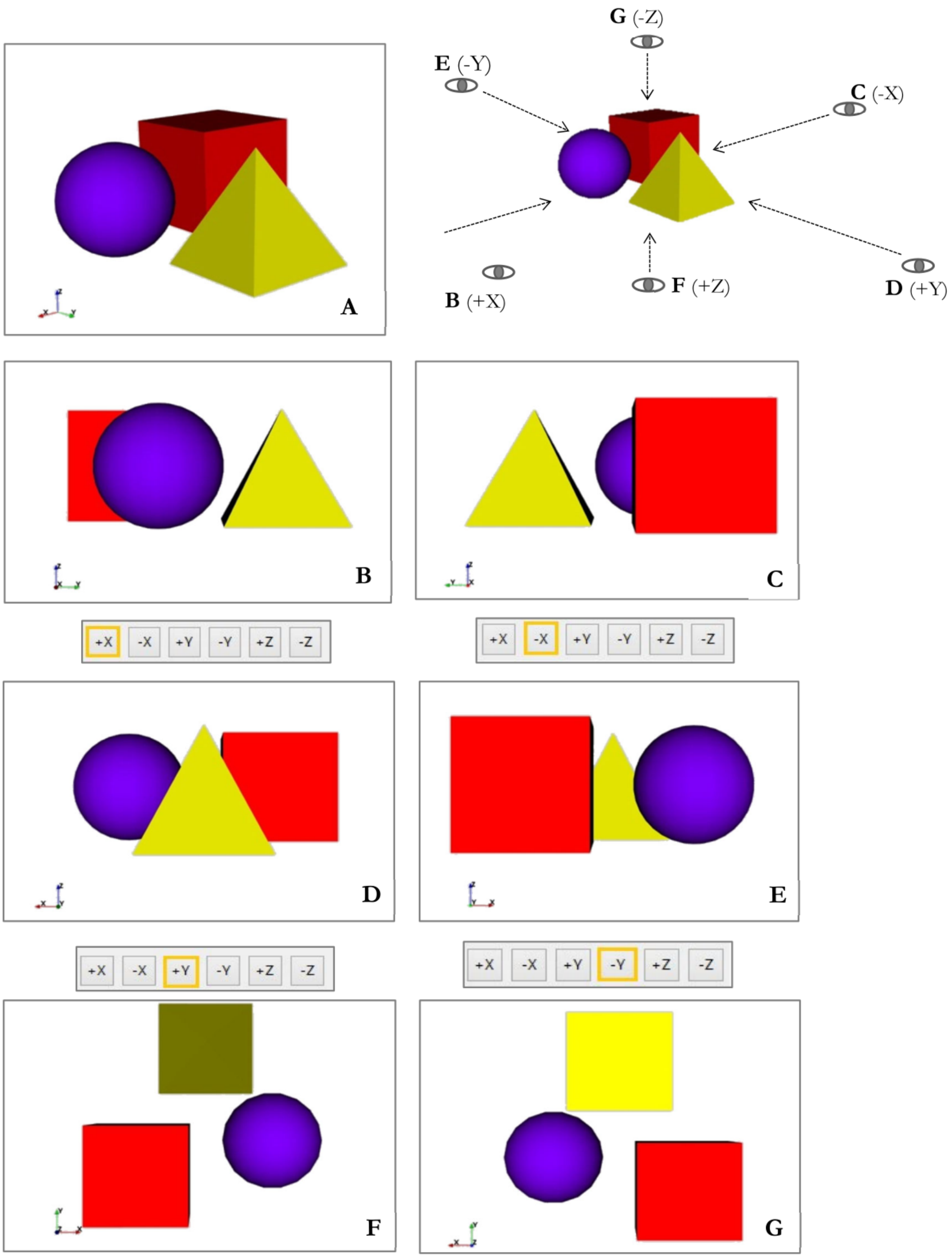


Figure 5.5: Different points of view of objects along one of the 3 axes

In A view from a casual point of view, in B view from positive X axis (+X), in C view from negative X axis (-X), in D view from positive Y axis (+Y), in E view from negative Y axis (-Y), in F view from positive Z axis (+Z) and in G view from negative Z axis (-Z)

5.4 Auto Centre

Sets automatically the object at the centre of the scene, as shown in Figure 5.6.

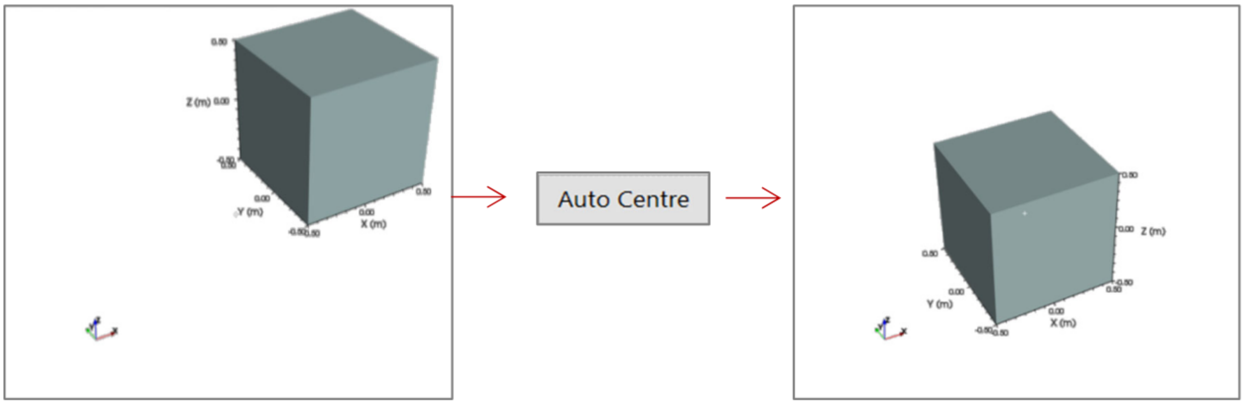


Figure 5.6: In A the object is in a general position, in B it is at the centre of the scene after the application of Auto Centre tool

5.5 Perspective

Switches from a perspective view to an orthogonal view (Figure 5.7). The red dashed lines helps to appreciate the difference between the two approaches to visualize the same object.

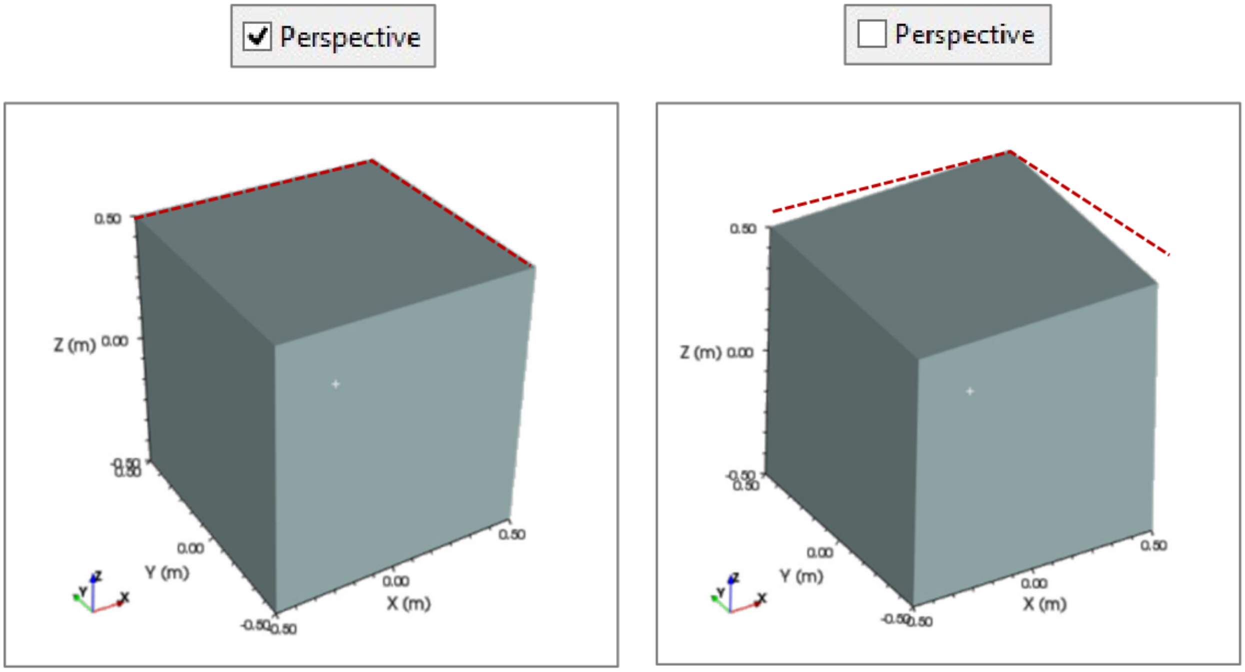


Figure 5.7: In A prospective view, in B orthogonal view

5.6 Azimuth and Elevation

They are an alternative to placing the object in the scene through the direct action of the mouse. It is possible to set specific values of azimuth and elevation. Starting from the point of view "+Y" (Figure 5.8 A), the value of the *Azimuth* determines the angle of rotation in horizontal, so around the Z axis (simulating a view of the object from the right with positive values, Figure 5.8 B and from the left with negative values, Figure 5.8 C). The *Elevation* value determines the angle of the vertical rotation, around the X axis (simulating a view for the top with positive values, Figure 5.8 D and from the bottom with negative values, Figure 5.8 E). Figure 5.8 F shows an example in which both values are different from zero (+35° for Azimuth and Elevation). In order to better appreciate the direction of the displacement of the cube, the logo of the software has been inserted in the face of the cube parallel to XZ plane.

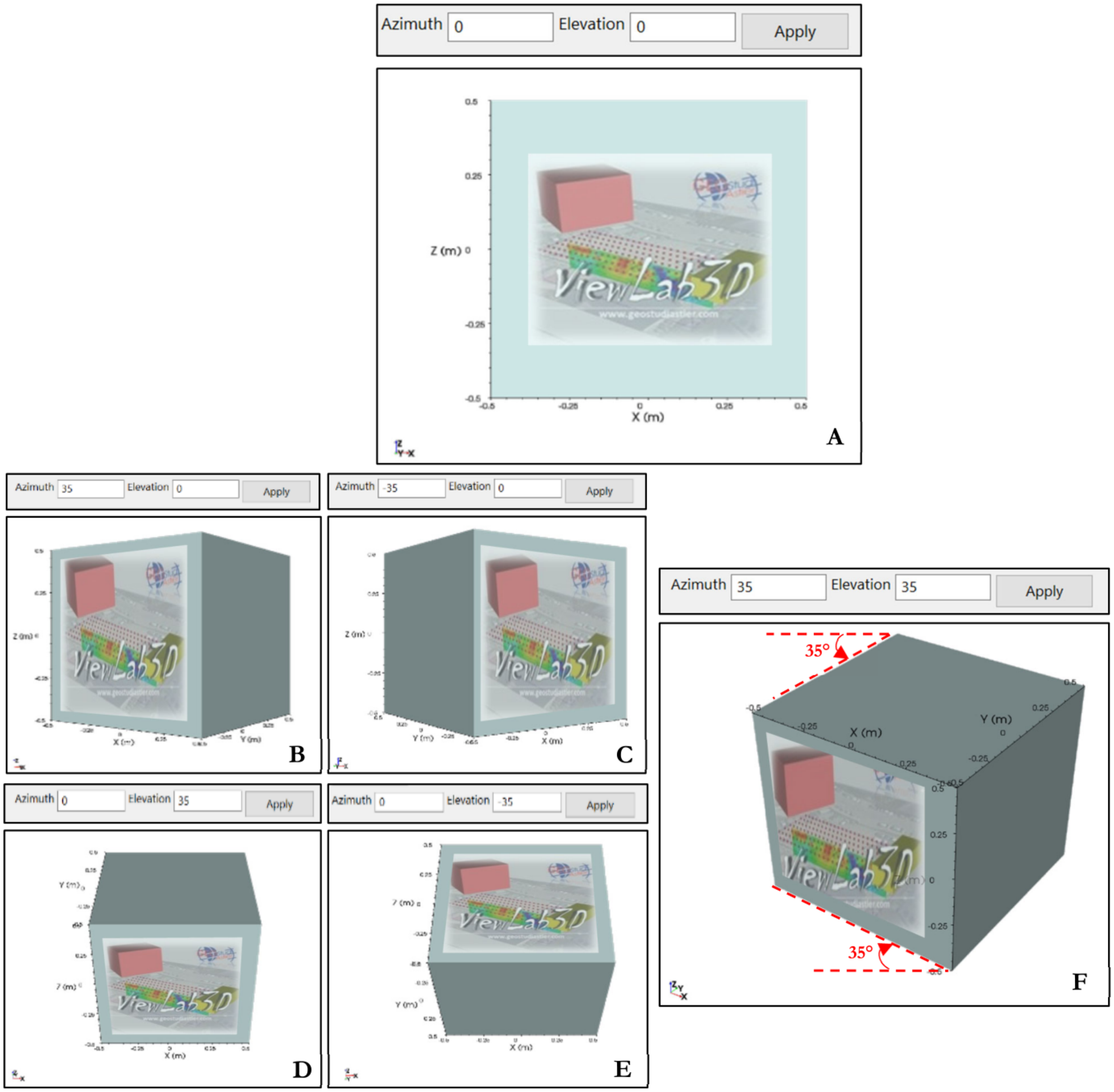


Figure 5.8: Different cases of Azimuth and Elevation values

To obtain similar results the camera can be moved also using the image buttons shown in Figure 5.9, discrete camera rotation will be performed at each click. It is also possible to set the *Angle* value to customize the rotation amount.

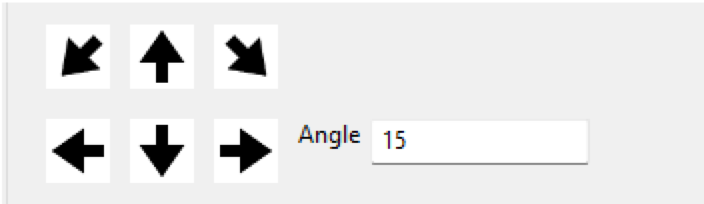


Figure 5.9: Discrete camera rotation

5.7 Favourite POVs

This tool allows to save some specific positions of the object in the 3D scene and to reload it later.

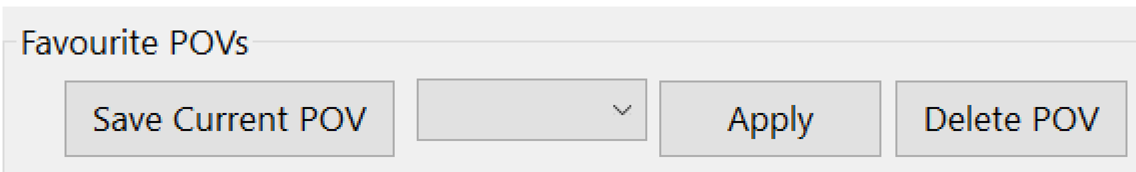


Figure 5.10: Point of View panel

To save the current view click on the "*Save Current Point of View (POV)*" button. It is possible to name a POV setting using the proper box (it is empty in the figure) and clicking *Apply*. Clicking on the "*Delete POV*" button deletes the selected Point of View.

5.8 Custom POV

Allows to set a specific Point Of View by inserting the desired coordinates X, Y, Z of the desired POV (*Point Of View*), FP (*Focal Point*), and N (*Normal*) in the proper table. The "**Get Current POV**" button updates the table to the values currently used in the 3D scene; it is possible to edit them to set the desired view mode (Figure 5.11).

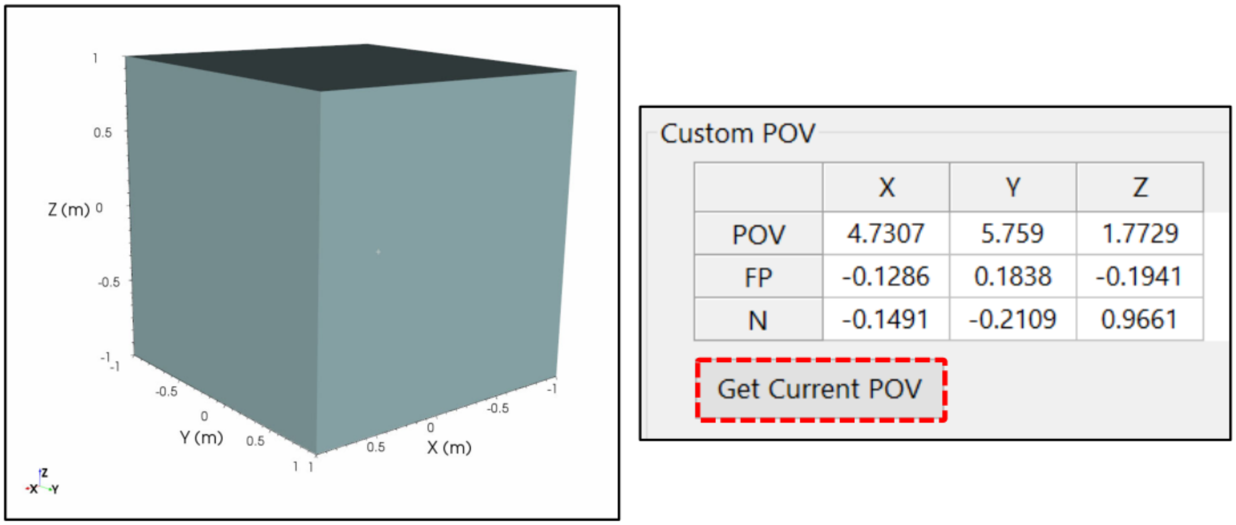


Figure 5.11: Autofill of table POV-FP-N in function of the position of the cube

CHAPTER

SIX

ORIENTATION TOOLS

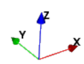


Figure 6.1: Orientation Tool

Through this panel (Figure 6.2) it is possible to set the properties of the object shown in Figure 6.1. It shows the orientation of the 3 axes (X, Y and Z) in the 3D space.

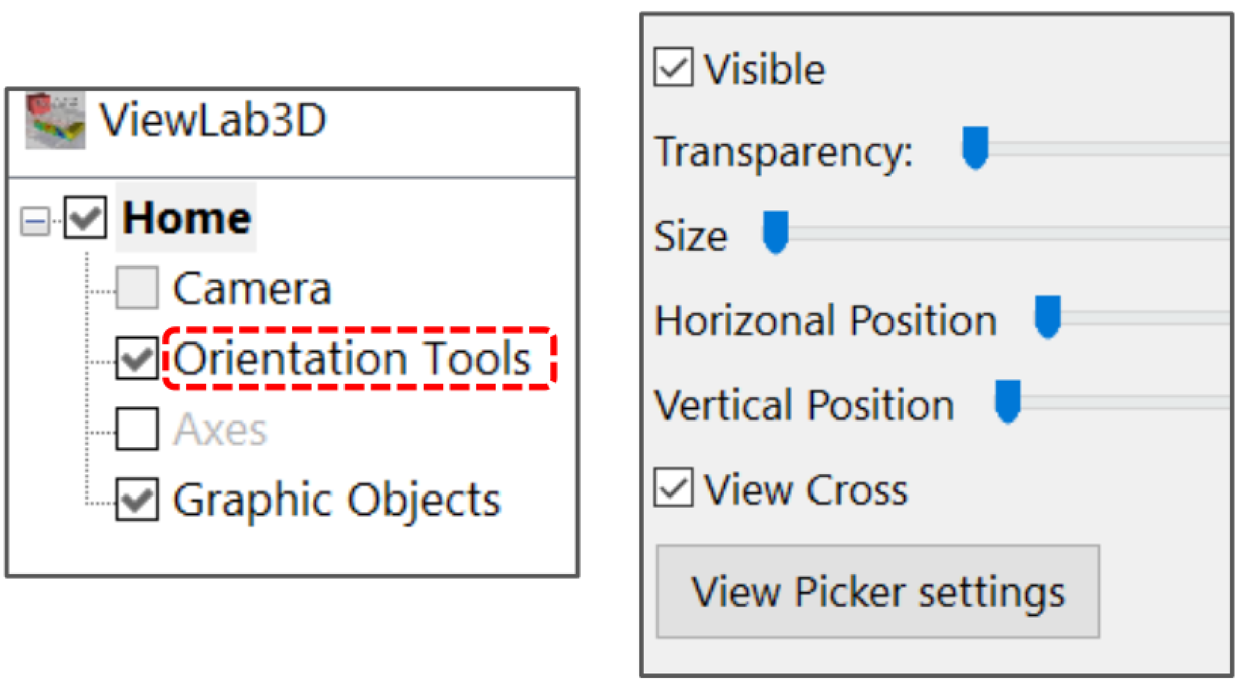


Figure 6.2: Orientation Tools panel

6.1 Visible

Allows to visualize the object or not, depending on if the box is checked.

6.2 Transparency

Allows the transparency of the object to be set. When the slider is totally at left the object is completely visible. When it is on the right it is not visible at all, totally transparent, and at intermediate positions it is more transparent as the slide goes towards the right, as shown in Figure 6.3.

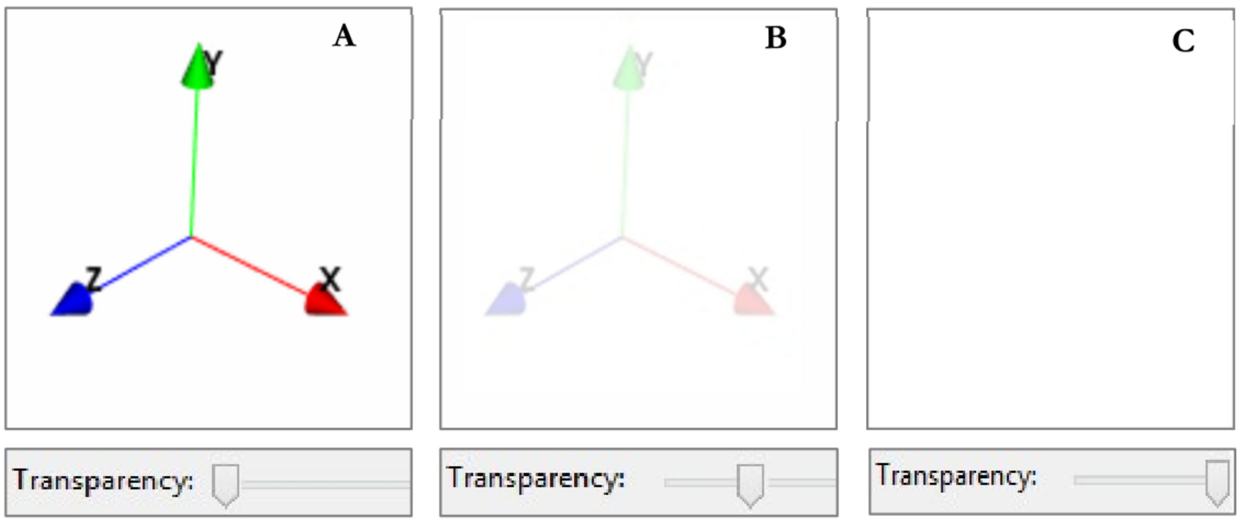


Figure 6.3: Different level of transparency of the object. In A the transparency is set at the minimum (in the detail the axes are completely visible), in B the transparency is set at an intermediate level (in the detail the axes are semi-transparent) and in C the transparency is set at the maximum level (in the detail the axes are not visible at all)

6.3 Size

Allows the size of the object to change. Between a maximum (slide at right) and a minimum (slide at left) depending on the position of the slider.

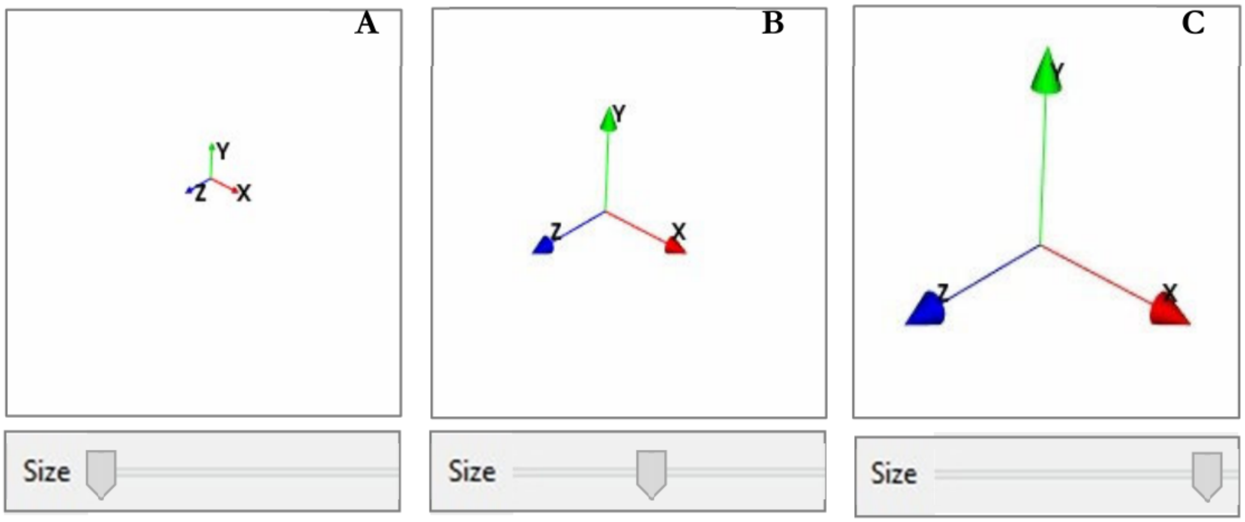


Figure 6.4: Different sizes of the orientation tool; in A minimum size, in B an intermediate size and in C maximum size

6.4 Horizontal Position

Allows to select the horizontal position of the Orientation Tool, between a maximum left side (slide at left) and a maximum right side (slide at right), with all the intermediate positions available, as shown in Figure 6.5.

6.5 Vertical Position

Allows to select the vertical position of the Orientation Tool, between a maximum lower side (slide at left) and a maximum upper side (slide at right), with all the intermediate positions available, as shown in Figure 6.5.

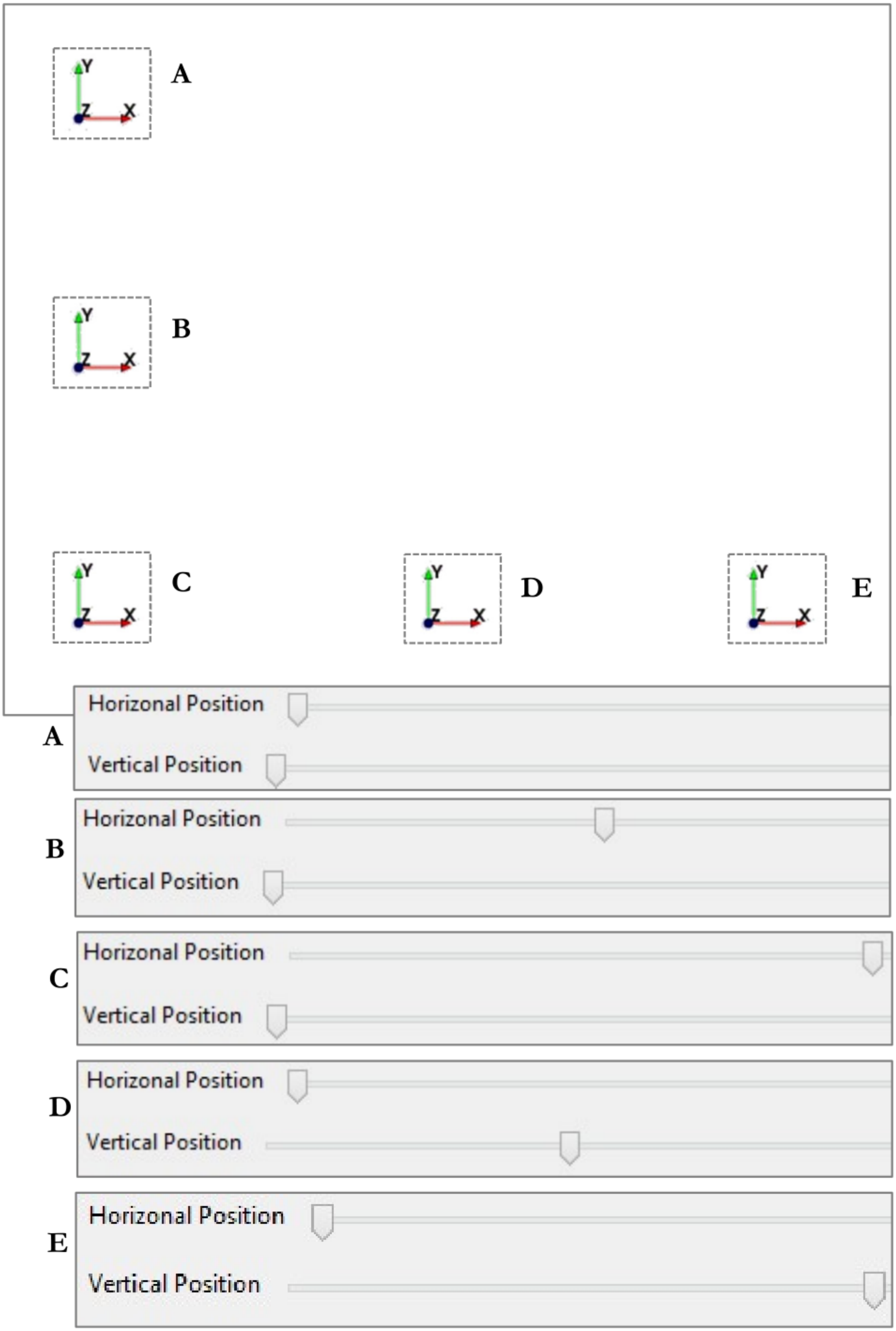


Figure 6.5: Different positions of the Orientation Tool Icon in the 3D scene panel

6.6 View Cross

If it is checked, the little white cross at the centre of the scene is visible, otherwise it is not.

6.7 View Mouse Indicators

If it is checked, the area (see Figure 3.4) clicked with the mouse in the main 3D area will be highlighted, other it will be not drawn. If the user wants to record the video during the exploration of the current 3D scene then disable this flag can be useful.

6.8 View Picker settings

Is a very useful tool, which we will recall many times in this manual, because it is useful in more than one case. The "picker" is a black point which follows the cursor when it moves in 3D space. It lets the user know the coordinates of the position occupied by the cursor in real time. To insert a point in the 3D scene and to save the coordinates of one picked point open the settings by clicking on the *View Picker setting* button, the proper panel will appear (Figure 6.6):

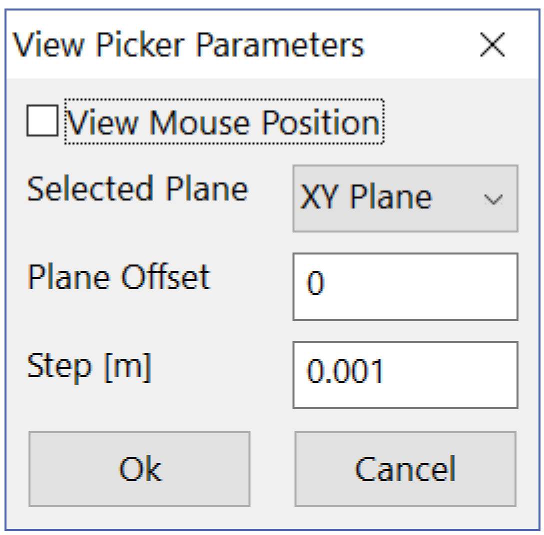
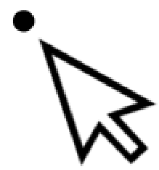


Figure 6.6: Picker Tool Panel

The same tool is also on the main toolbar of the 3D scene.

6.8.1 View Mouse Position

Is enabled when checked, a black point, which follows the cursor around, will appears in the 3D scene.



At the same time the X, Y and Z coordinates will appear at the bottom of the 3D scene. Clicking with the right mouse button in the 3D scene, while this tool is activated, the coordinates of the position occupied by the cursor in that moment are saved and the point remains fixed in the scene until a new point is picked. At the bottom of the 3D scene the stored coordinates are shown as "*Last Picked X, Y, Z*", together to other information like the distance between the two points or the angles between them.

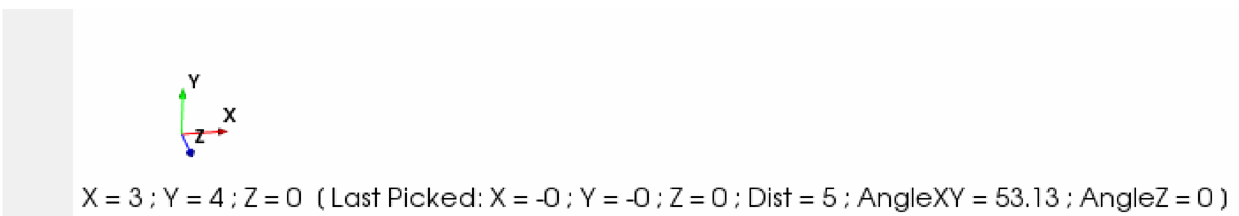


Figure 6.7: Mouse position

6.8.2 Select plane

Lets the user choose the plane on which the point will appears, through a pop-up menu as shown in Figure 6.8.

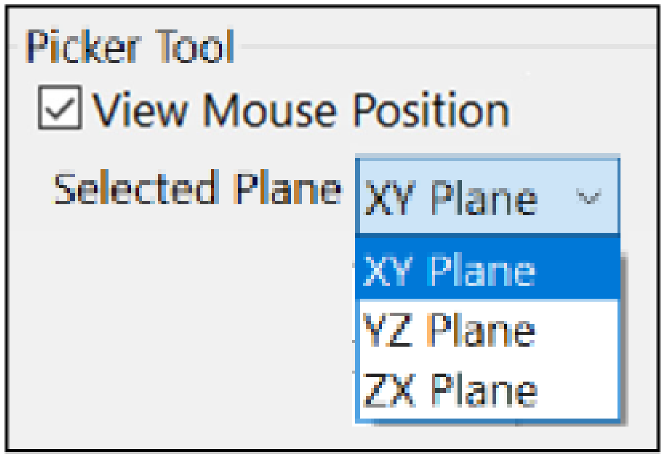


Figure 6.8: Picker Tool_Select Plane

In many cases it can be useful to disable the prospective during the mouse position analysis to avoid parallax problems; also an setting a view that is orthogonal to the picking plane used can help (see Camera options). To make the picking correctly, the plane chosen in this menu should match the view currently visualized in the 3D scene.

Selecting the "XY Plane", "YZ Plane" and "ZX Plane" the main 3 orthogonal planes will be used to pick the mouse position. Selection the last option, "Objects", the picking process is performed using the 3D object the mouse cursor is hovering over, providing a generic location independent from the three main planes.

6.8.3 Plane offset

The *Picker Point* moves on a 2D plane, the third dimension is set to 0 by default. Through this tool the user can change this value and to define a value for the third dimension, the position of the point in the direction perpendicular to the selected plane.

In the following example (Figure 6.9), a cube is inserted in the scene, with a Zmax of 3m; the selected plane is XY. So the X and Y coordinates are readable directly on the axes. The plane offset in this case determines the Z coordinate, which is 0 in one case and 3 in the other. To make the point visible, the Z of the picker must be equal or higher than the Z coordinate of the examined object (otherwise it goes under the object and it is not visible).

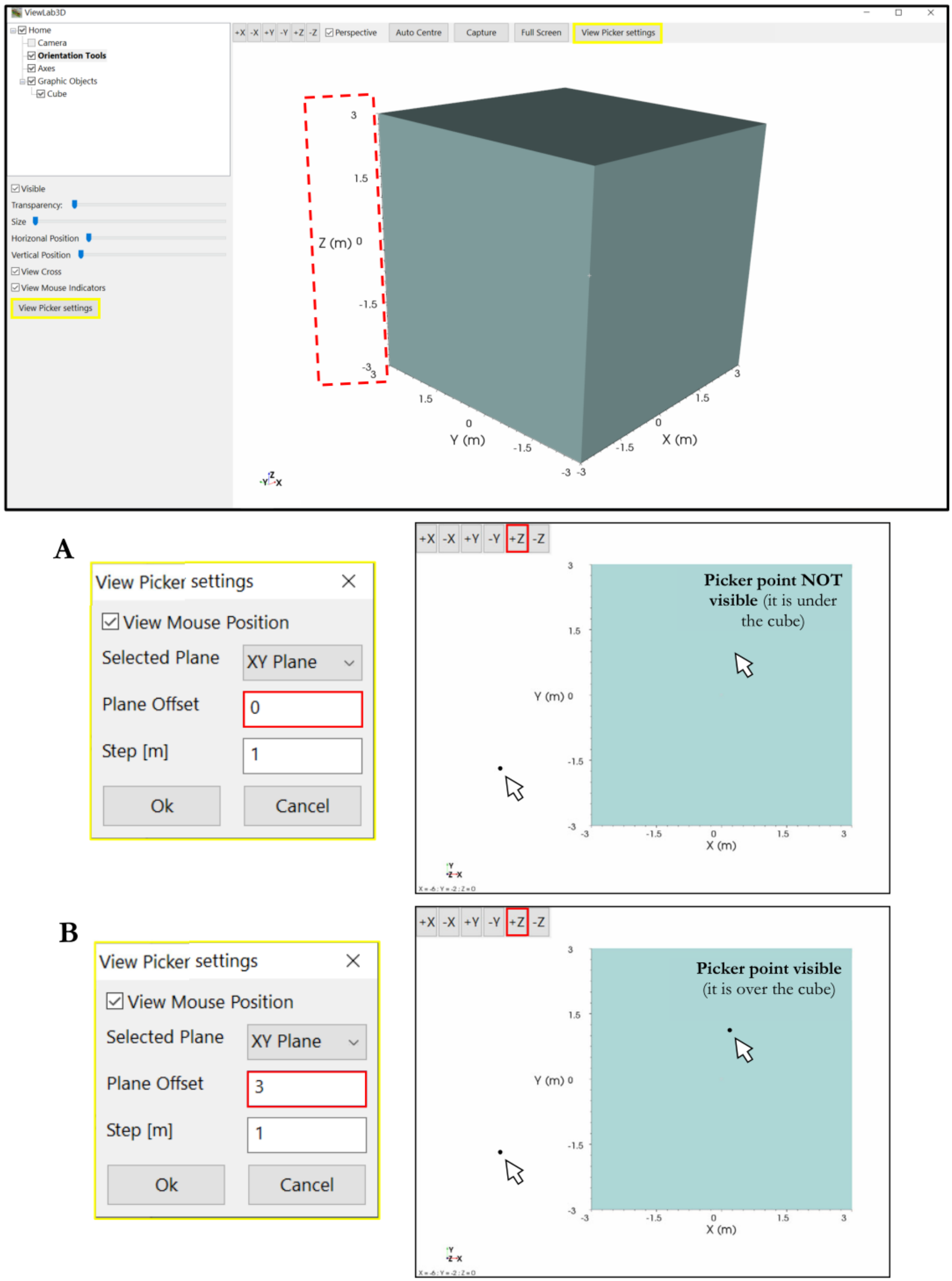


Figure 6.9: Plane offset of Picker Point

6.8.4 Step

This tool helps to insert a point at defined values of X and Y coordinates or a sequence of points at a certain interval distance. The number set in this box is the step with which the point moves in the 3D space. To make it move smoothly, set this number at very low value (0.001m by default). To make it move a defined interval value (e.g. 0.5m in Figure 6.10), write this number in the proper box and the point will move from its position to the next just when the cursor will be moved at least the defined distance.

In the following example (Figure 6.10), in A the step is 0.001m, so moving the cursor from X = 0 to X = -0.5, in the direction suggested by the red arrow, the pointer follows the cursor smoothly and in B the step is set to 0.5 so the point moves directly from X=0 to X=-0.5, in one single step.

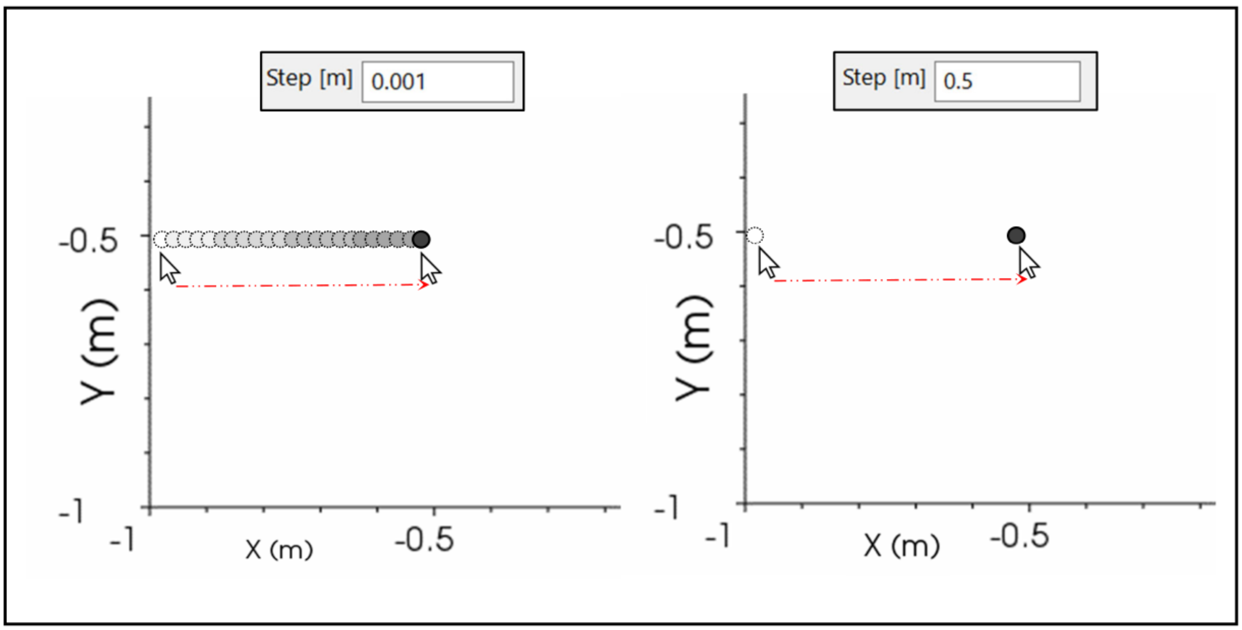


Figure 6.10: Different Steps of Picker Point

6.8.5 Mouse Position

Once the Picker Tool is set, it can be used to read the coordinate in real time, to insert electrodes through the direct interaction of the user with the scene (see Continuous mouse append), or simply to obtain the coordinates of a specific point in the 3D scene. This is the case in the following example where an object was inserted in the 3D scene (house). The selected point of view is +Y, so it is necessary to select the XZ plane in the picker point setting panel. It is advisable to uncheck the perspective view. Clicking with the right mouse button on the top right of the house, the proper coordinates are saved and they appear at the bottom of the screen (Figure 6.11).

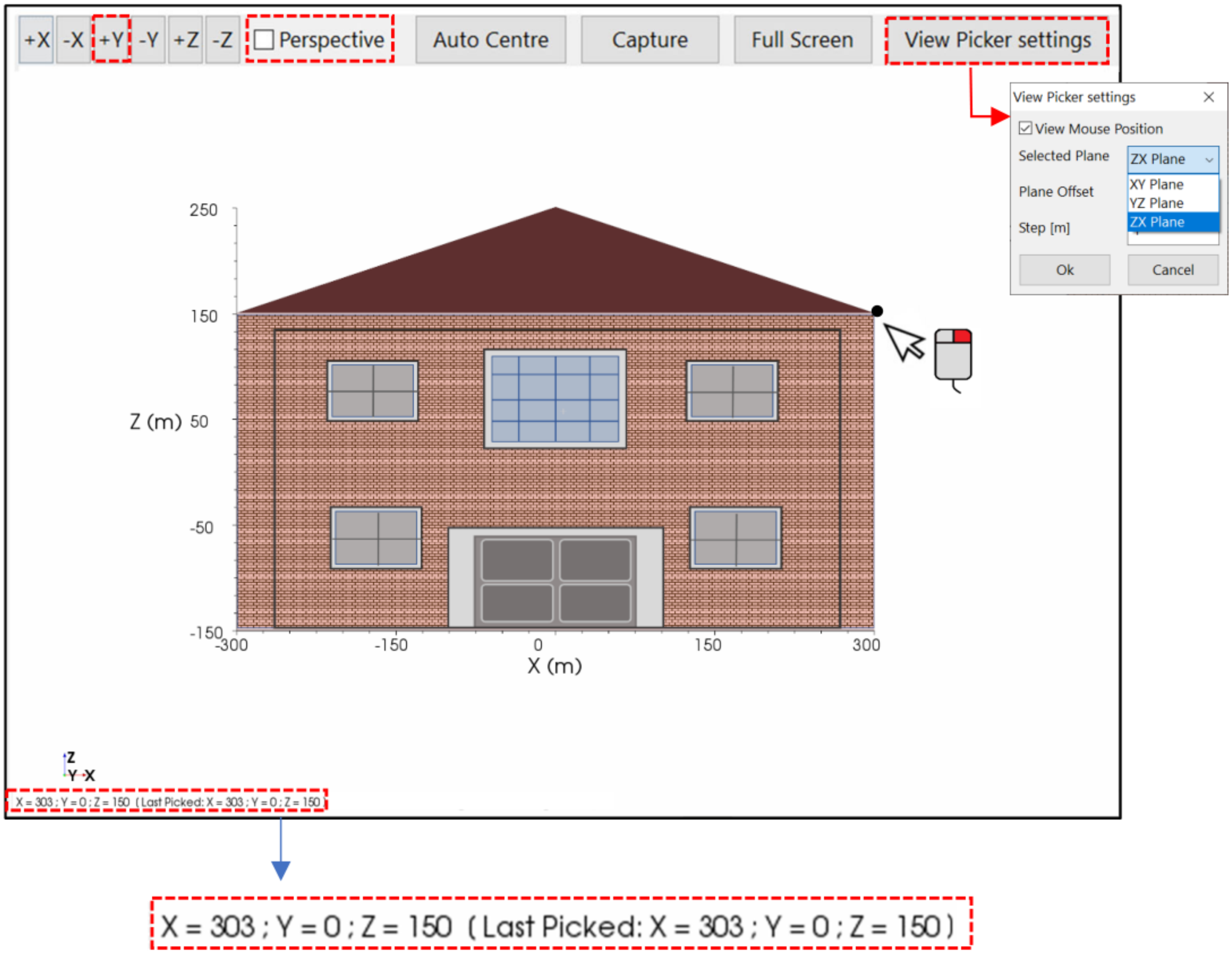


Figure 6.11: Example of coordinates obtaining

CHAPTER

SEVEN

AXES

Through this panel it is possible to set the property of the axes shown in the 3D scene (Figure 7.1).

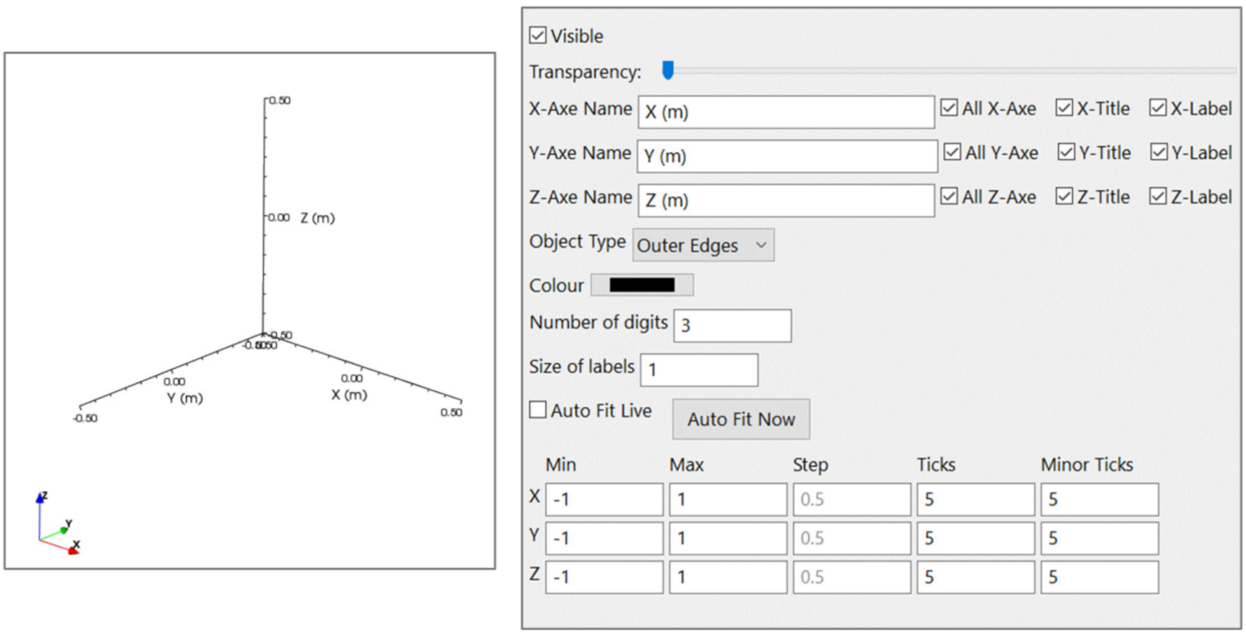


Figure 7.1: "Axes" panel tools

7.1 Visible

Allows to show the axes or not, depending on checking the box.

7.2 Transparency

Allows the user to set the transparency of the axes. When the slider is totally at left the object is completely visible, when it is on the right it is not visible at all. At intermediate positions it is more transparent as the slider goes towards the right.

7.3 X Y Z Axis name

Allows to rename each axis (X(m), Y(m), Z(m) are the default names), as it is shown in Figure 7.2.

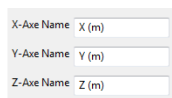


Figure 7.2: Detail of “Axes” panel

7.4 All X Y Z Axis

Sets the visibility of each single axis, independently from the others two., e.g. in the Figure 7.3 the Z axis is visible in the first and not visible in the second case.

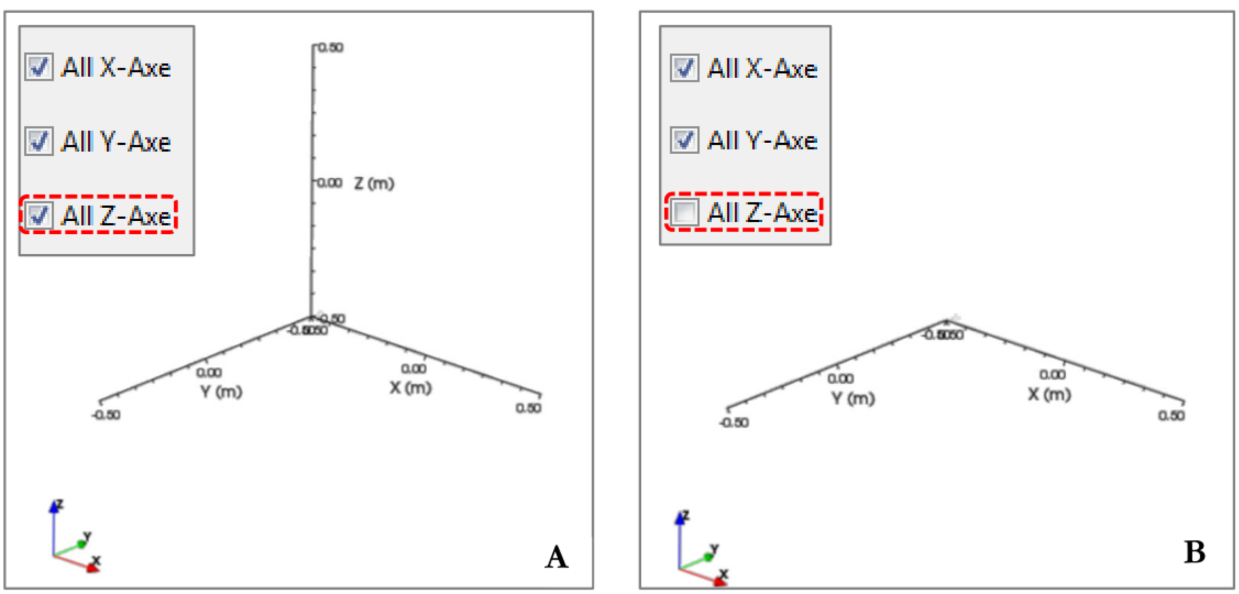


Figure 7.3: In A all the three axes are visible, in B the Z axis is not visible

7.5 X Y Z Title

Sets the visibility of each single axis title independently from the others two, e.g. in Figure 7.4 the title of the X axis is visible in the first and not in the second case.

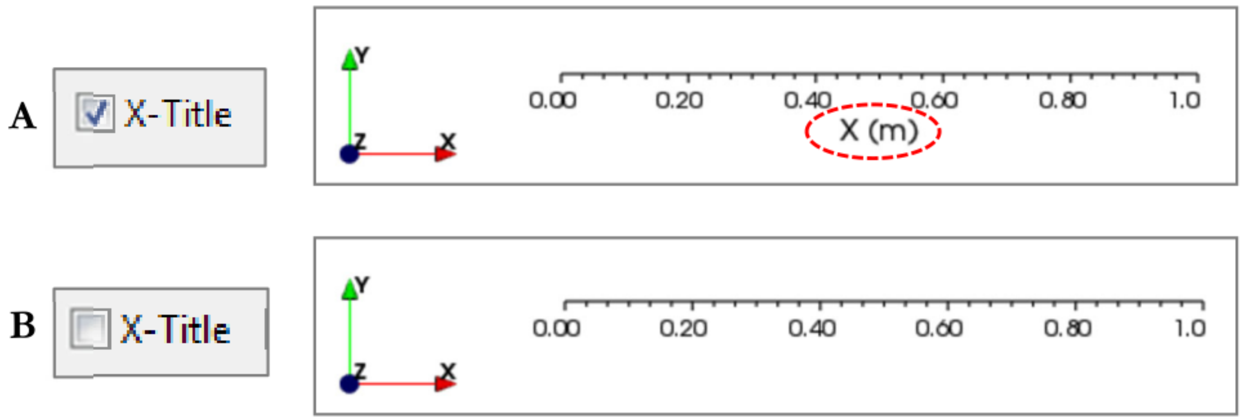


Figure 7.4: In A the title is visible, in B not

7.6 X Y Z Label

Sets the visibility of each axis labels independently from the others two, e.g. in Figure 7.5 the labels of the X axis are visible in the first and not in the second case.

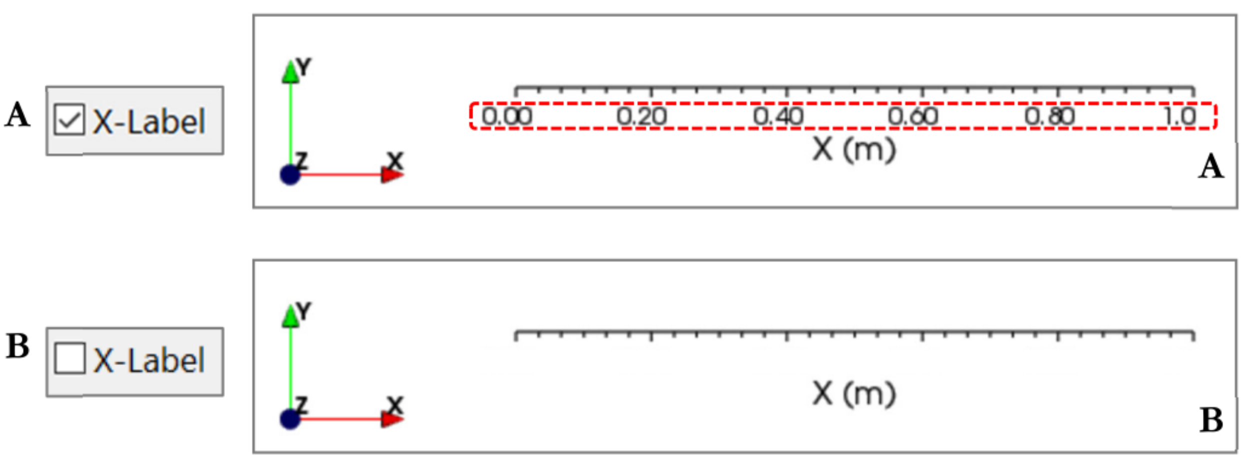


Figure 7.5: In A all the X axis value (labels) are visible, in B they are not visible

7.7 Object Type

This pop-up menu sets the behaviour of the axes when the object is moved, letting the user choose between three options:

- *Outer Edges*: This puts the axes along the outer edges of the object in the scene (the cube in the example) and they maintain this position even if the object is moved (they are not linked to the object), as shown in Figure 7.6.
- *Closest Triad*: This puts the axes along the nearest triad from the point of view of the observer and they maintain this position even if the object is rotated (they are not linked to the object), as shown in Figure 7.6.
- *Fixed*: This puts the axes in a fixed triad (starting from the Xmin, Ymin, Xmin) which is hooked to the object; if the object is rotated the axes rotates with it (Figure 7.6).

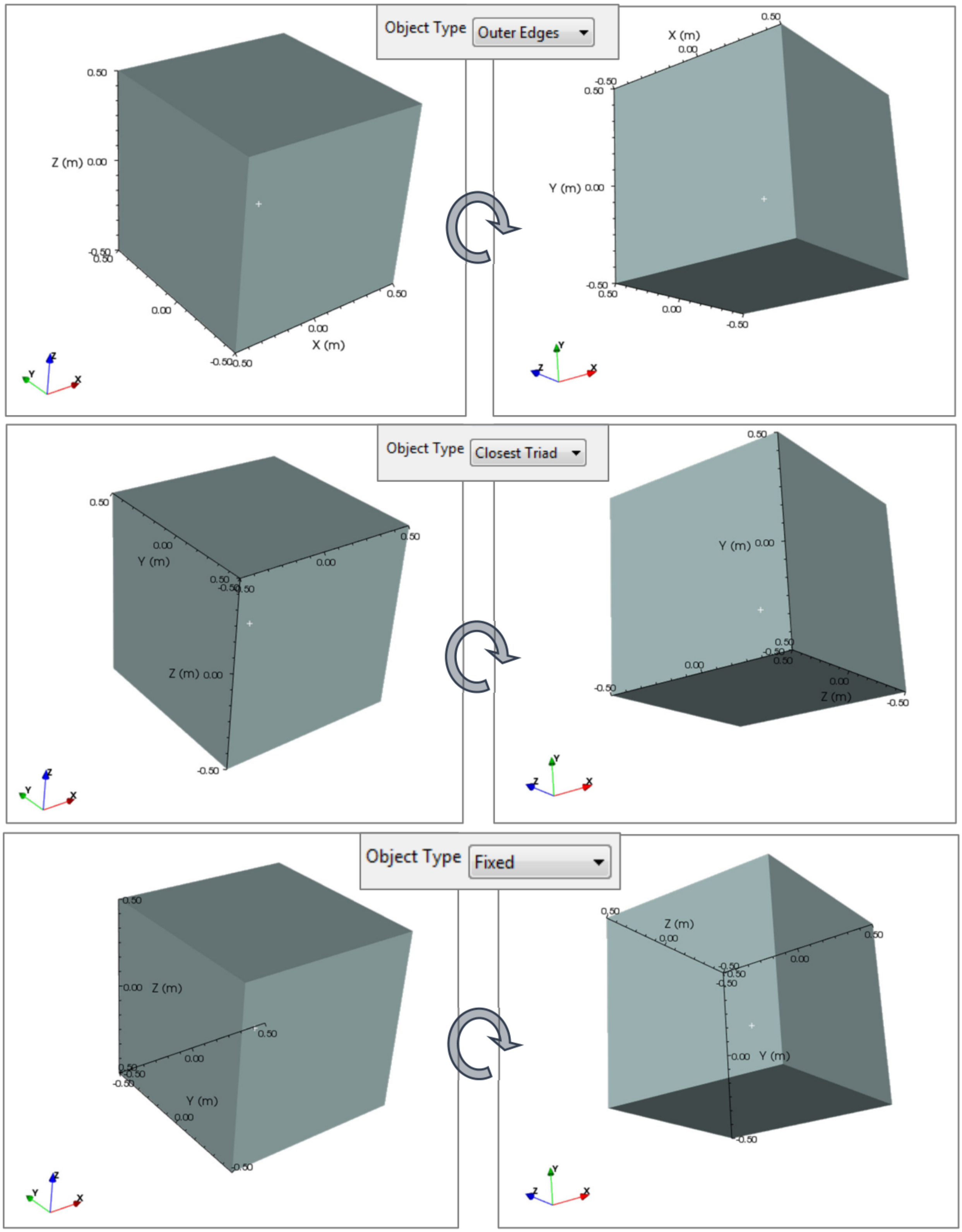


Figure 7.6: Before (left columns, A1, B1, C1) and after (right columns, A2, B2, C2) a rotation of the object. Only in the "closest Triad" case the axes do not follow the cube

7.8 Colour

Allows to change the colour of the axes; some examples are shown below (Figure 7.7).

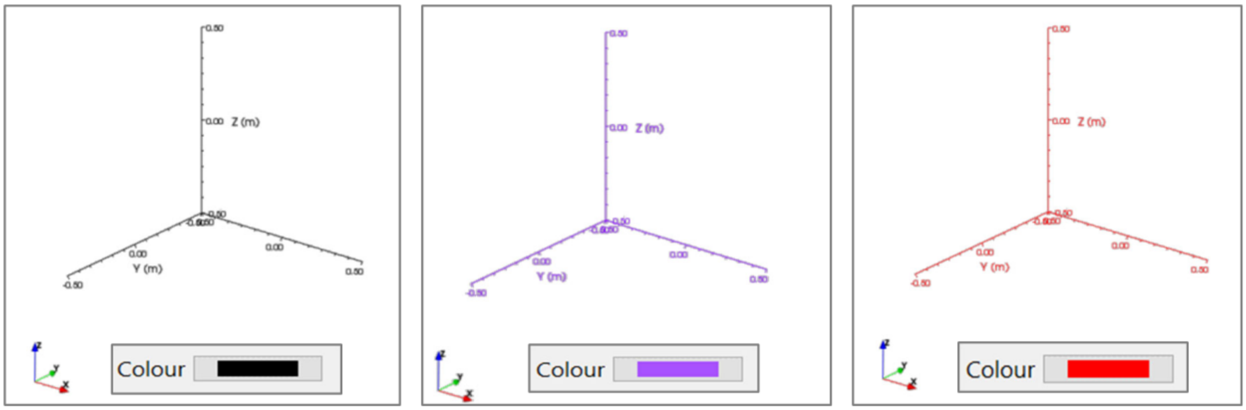


Figure 7.7: Different choices for axes colours

7.9 Number of digits

Allows to set the number of decimals of the axes labels shown. It is possible to specify the value both graphically, with the “slider”, or numerically, using the text box.

As an example, in the following figure the number of digits is set to 2 and 4, respectively.

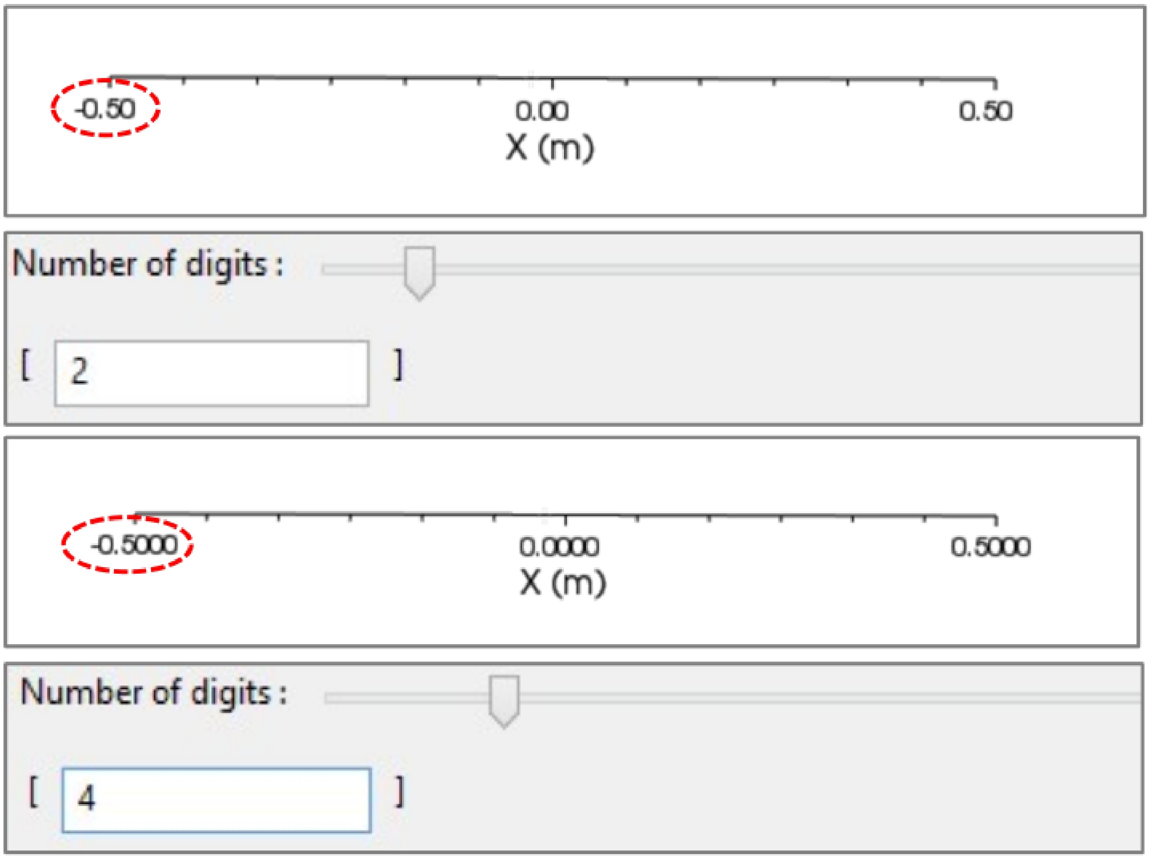


Figure 7.8: Example of different numbers of digits

7.10 Size of labels

Selects the font size of the numbers near the axes, specifying graphically the value through the "slider" or by numerically editing the text box (Figure 7.9).

Figure 7.9: Example of different labels sizes

7.11 Auto Fit

- **Auto Fit Live:** If this is selected, the software automatically and continuously adjusts the size of the axes according to the size of object shown in the 3D figure (Figure 7.10).
- **Auto Fit now**: This function has the same functionality as the checkbox above, but it works only one time when the button is pressed (Figure 7.10).

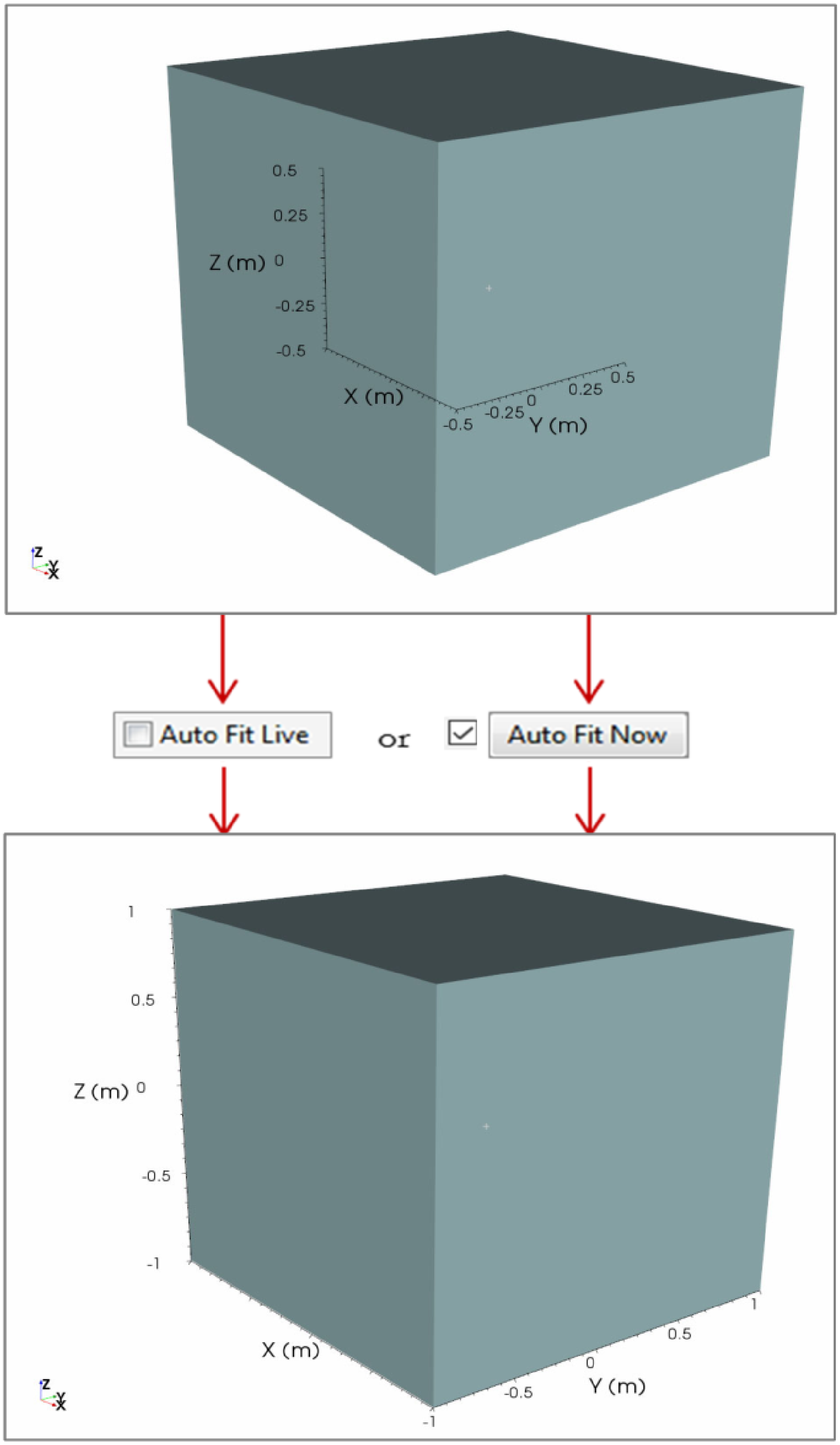
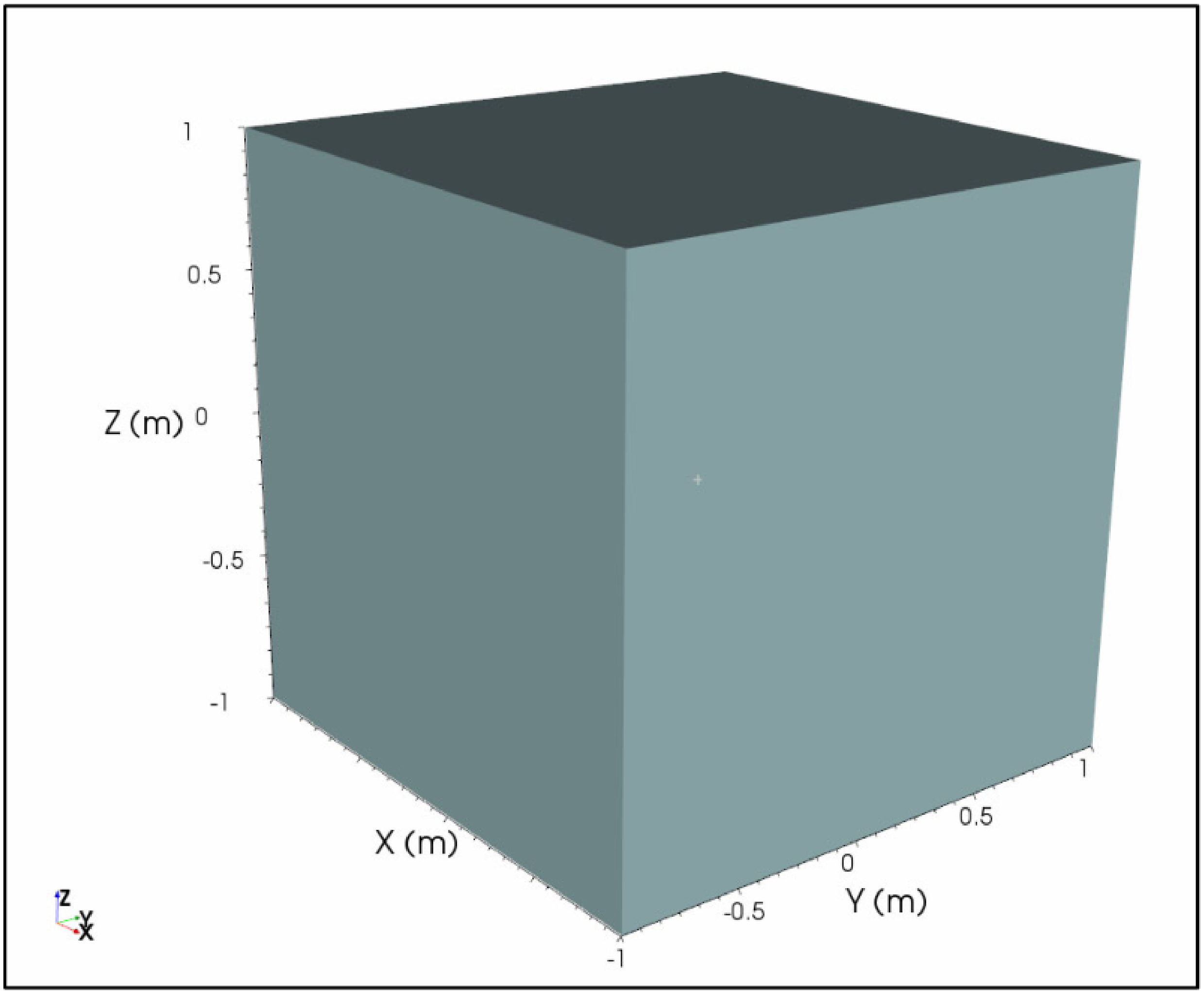


Figure 7.10: In A the axes does not fit the object, in B they follow the object because “Anti Fit live” or “Auto Fit Now” tool is active

7.12 Table

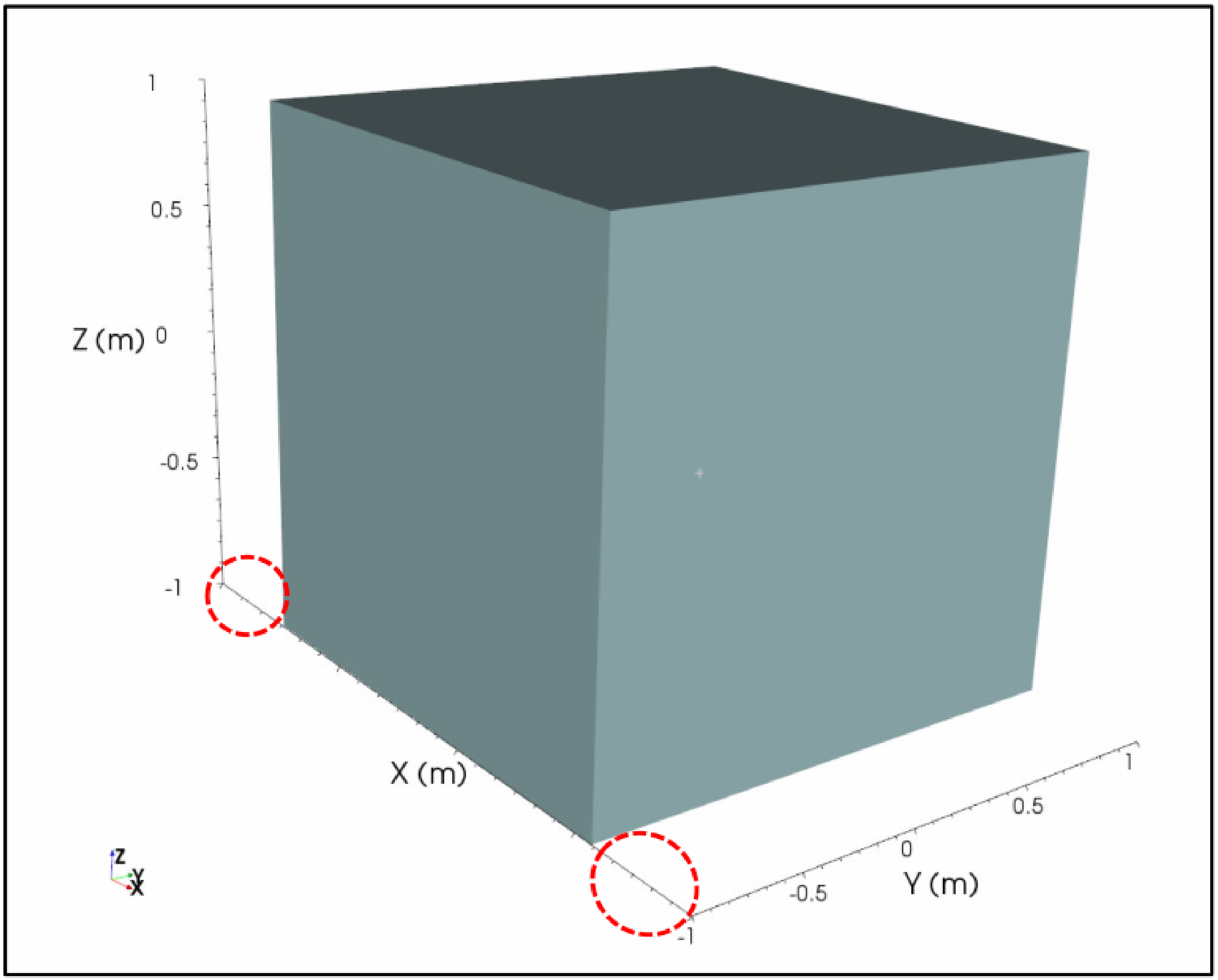
The **table** in the lower part of the screen allows to set the dimensions of the 3 axes and the way to display them. The following examples shows some customization possibilities. In this case (Figure 7.11), the axes are set in automatic way, through the *Auto Fit Live* tool, so the minimum and maximum dimensions of the axes are the same as the object.



	Min	Max	Step	Ticks	Minor Ticks
X	-1	1	0.5	5	5
Y	-1	1	0.5	5	5
Z	-1	1	0.5	5	5

Figure 7.11: Example of axes with “Auto fit Live” tool activated

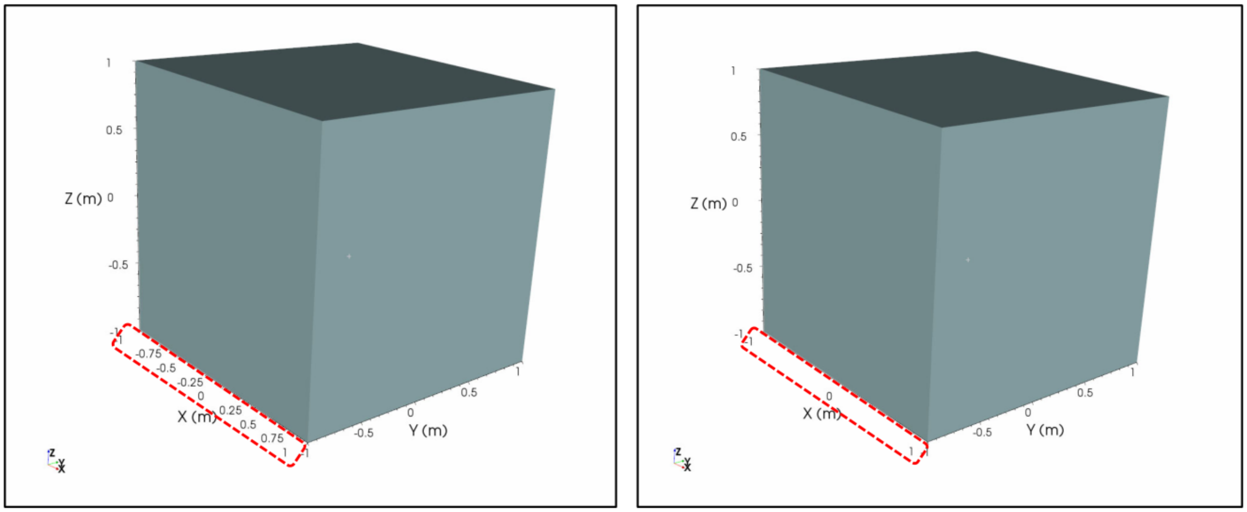
It is possible to change the axis length, e.g. in Figure 7.12 the X axis starts from -1.5 and ends at +1.5, while the object size is still between -1 and +1.



	Min	Max	Step	Ticks	Minor Ticks
X	-1.5	1.5	0.75	5	5
Y	-1	1	0.5	5	5
Z	-1	1	0.5	5	5

Figure 7.12: Example of axis longer than the object

It is possible to change the features of axes labels as well. It is possible to set the *Step* (how many digits to skip between one label and the next) or the *Ticks* (how many labels to display). The two quantities are dependent on each other, so setting one of them ERTLab *Studio*/ViewLab3D will calculate automatically the other one (greyed out numbers in the table). In the example in Figure 7.13 the X axis *Step* is set at 0.25 and the *Step* of Y and Z axis are not modified (they are still at 0.5); the *Ticks* are automatically calculated, and it is 9. On the right hand the *Ticks* of the X label are set to 3 instead, still 5 for the Y and Z axis. The *Step* is now automatically calculated, and it is 1.



	Min	Max	Step	Ticks	Minor Ticks
X	-1	1	0.25	9	5
Y	-1	1	0.5	5	5
Z	-1	1	0.5	5	5

	Min	Max	Step	Ticks	Minor Ticks
X	-1	1	1	3	5
Y	-1	1	0.5	5	5
Z	-1	1	0.5	5	5

Figure 7.13: Examples of different Step (A) and Tick (B) of the X axis

CHAPTER

EIGHT

GRAPHIC OBJECTS

Through this panel the user can insert various types of objects in the 3D scene.

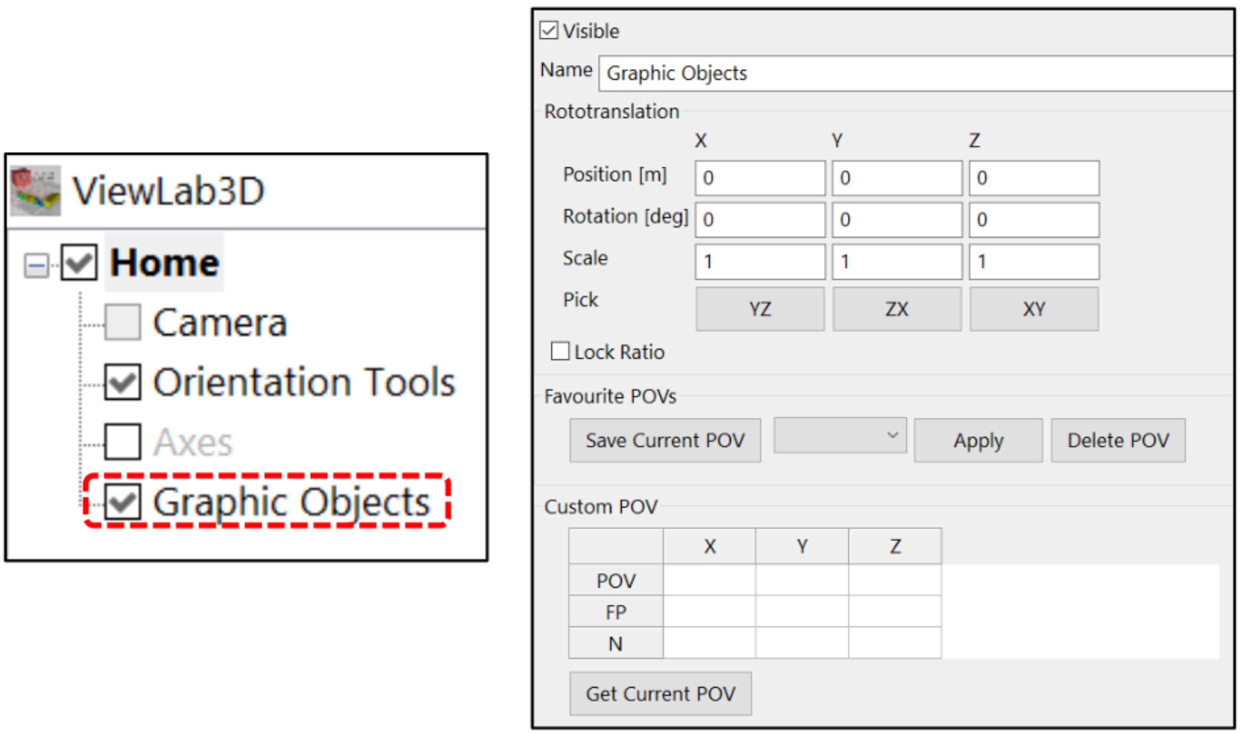


Figure 8.1: Graphic Objects panel

Using the "*Visible*" property the user can check the visibility of all the objects placed in the scene. Each inserted object is automatically positioned at the origin of the three axes (0,0,0). Through the table X-Y-Z it is possible to change this, making a *Translation* (to translate the origin to a different point in space), a *Rotation* (rotation in degrees around each axis) or using a *Scale* tool (useful if there is the necessity to emphasize one dimension or to work with many objects of different dimensions). Clicking with the right mouse button on the **Graphic Objects** submenu, it is possible to load an object saved in a personal folder, using the *Load* button, or to insert one of the available objects through the *Insert Object* button. In this case a panel with a list of tools will appear, as shown in Figure 8.2.

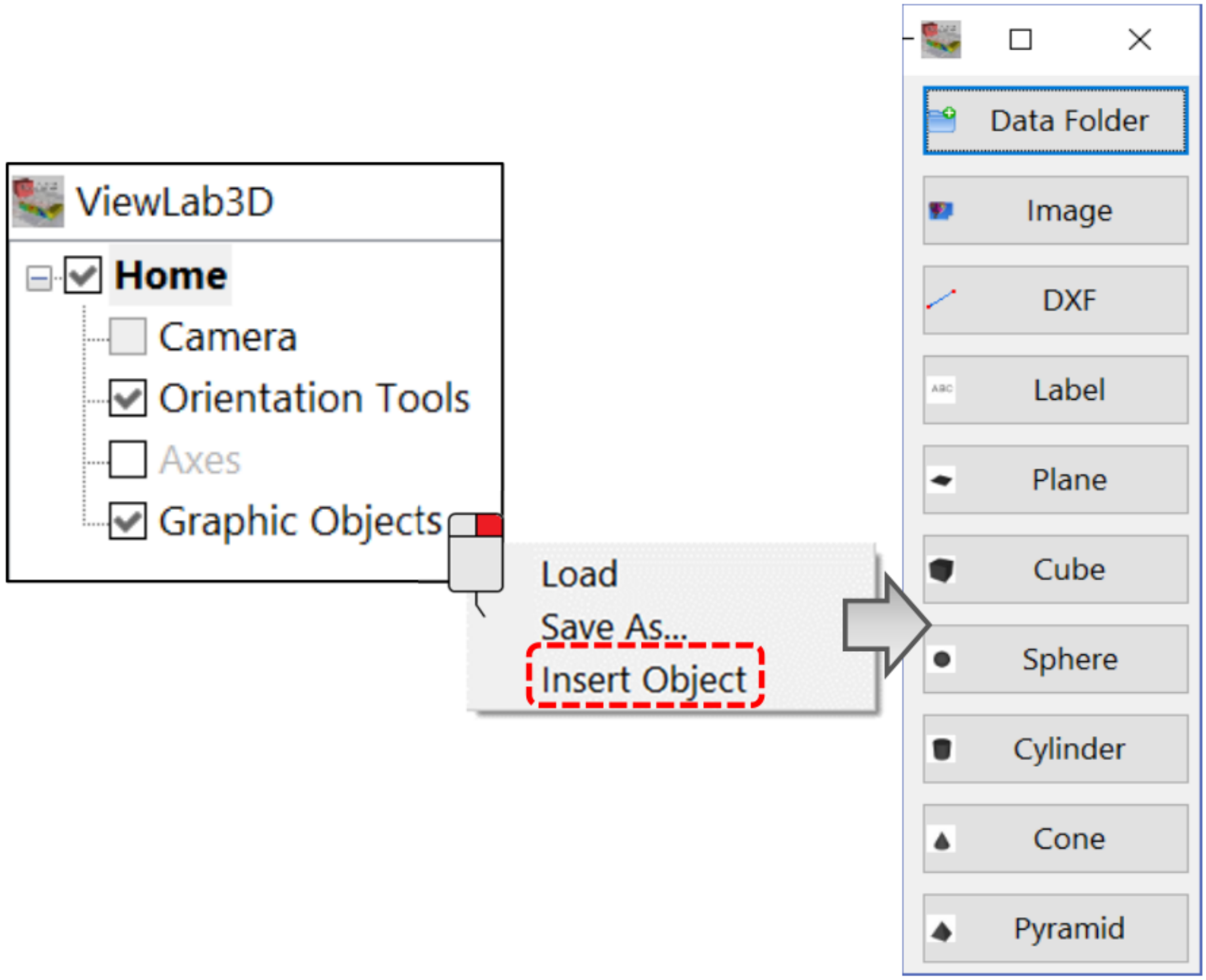


Figure 8.2: Objects available for the insertion in 3D scene

8.1 Data Folder

It is possible to create and to save some custom objects using a combination of the other simple objects. Insert a new “Data Folder” through *Insert Object*. A new node “Graphic Object” will be added to the tree menu (Figure 8.3). This method is a way to group together more objects and manage them with a simple click (for example they can all be shown/hidden with one click, or they can moved just editing a single rototranslation panel).

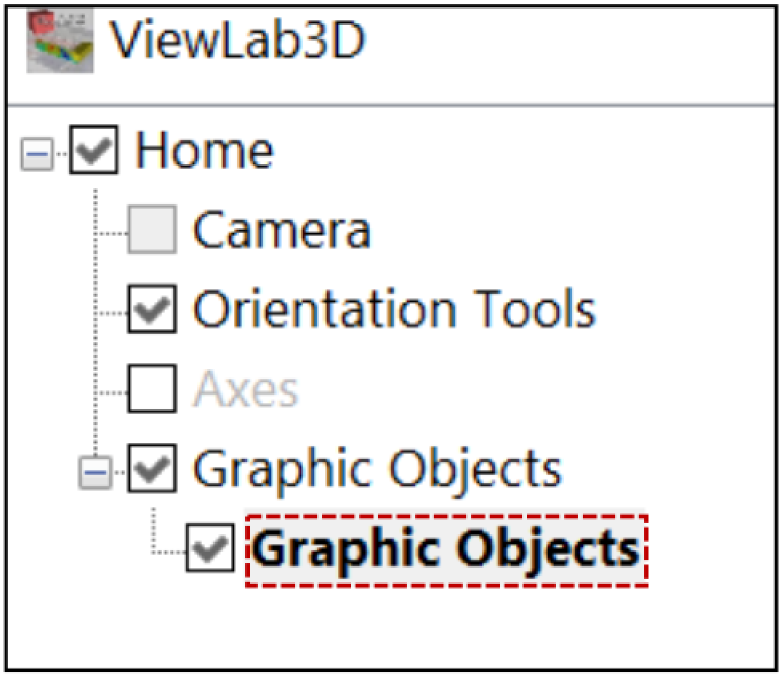


Figure 8.3: Graphic Object node added to the menu

In the following example a house is created using one cube, one pyramid and 2 images. The *Image* is a photo of a front of a house; in this example it is used two times, to cover two sides of the cube. The setting of each object used to make the house as shown in Figure 8.4.

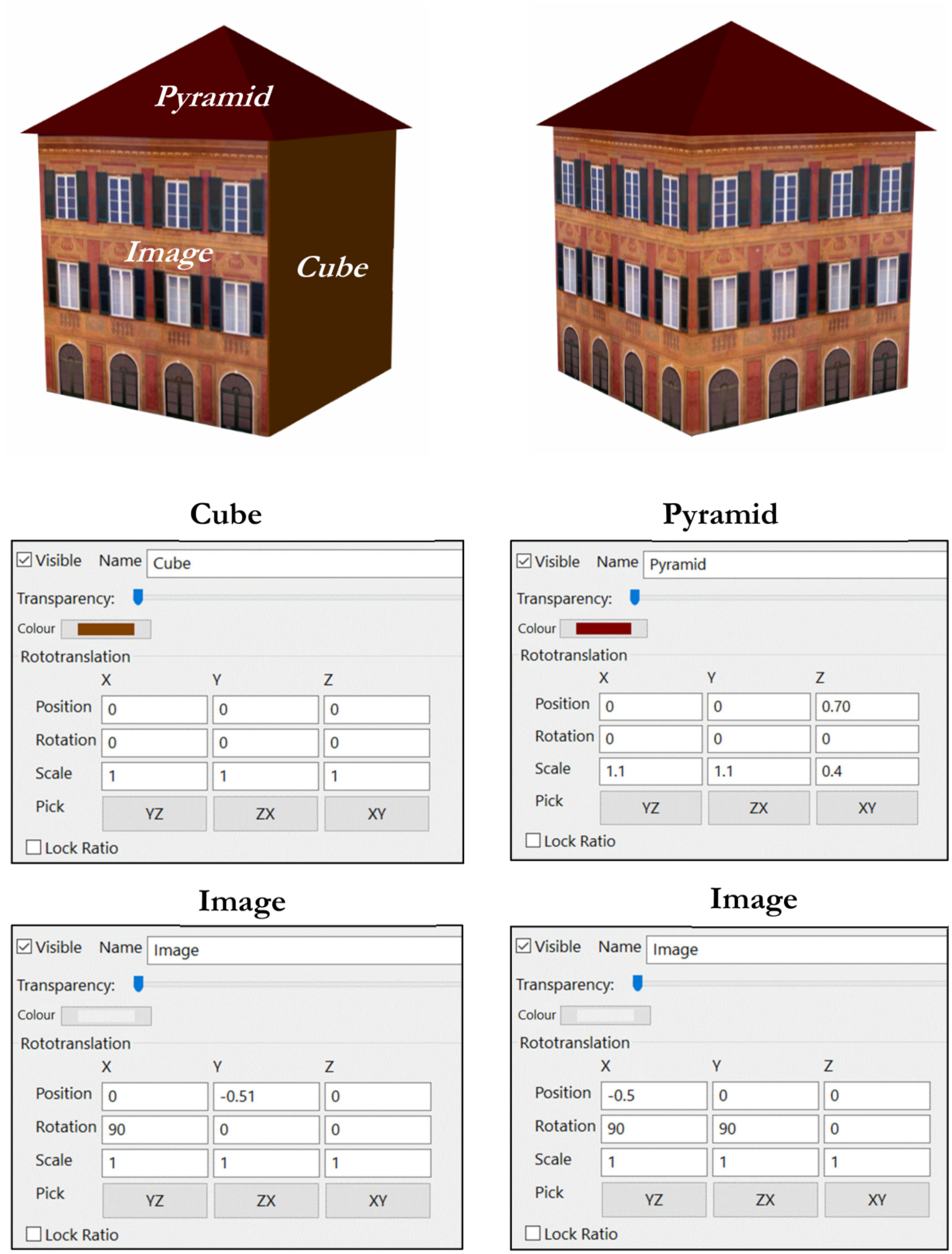


Figure 8.4: Example of creation of custom object

Once the new object is created, it is possible to give to it an appropriate name through the setting panel (“House” in this example) and to save it through the “Save As” menu option which appears by clicking with the right mouse button on the object name (Figure 8.5).

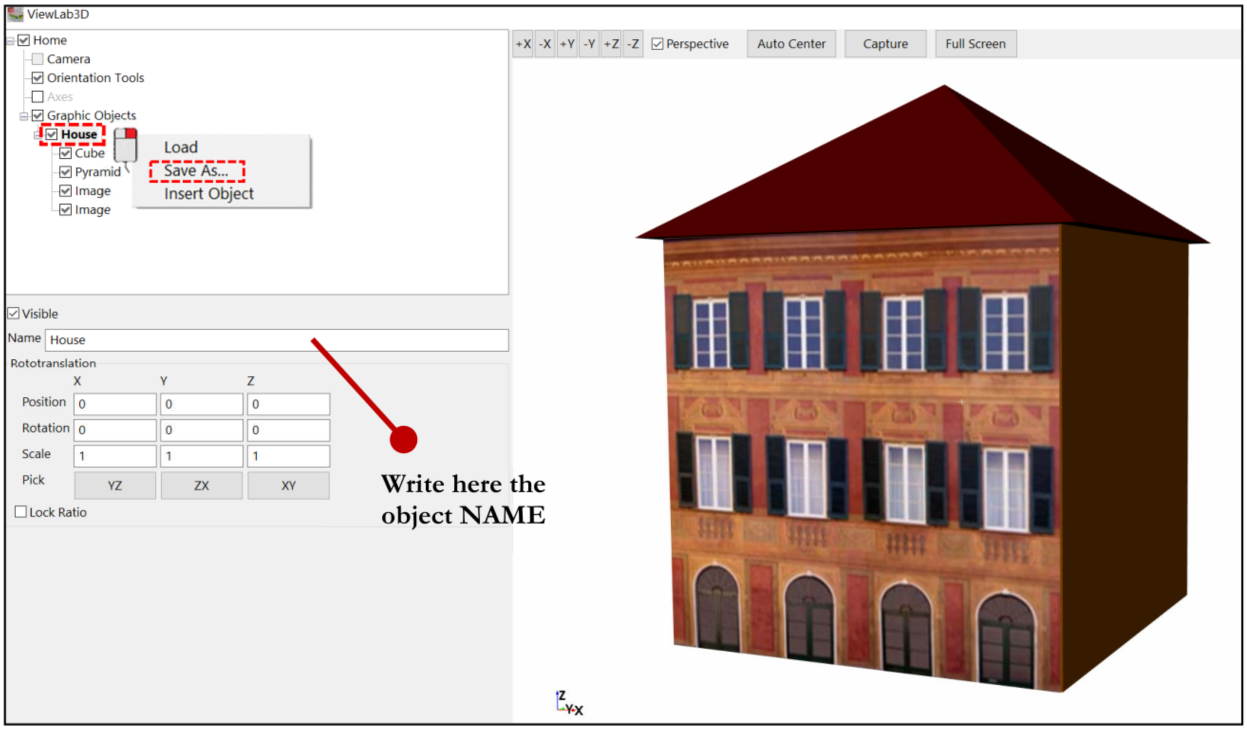


Figure 8.5: Saving new created object

This way the user can load it again for further projects using the appropriate menu option “*Load*” in the object menu. This is a very useful instrument, which lets to customize the project with personal photos of the survey area (for example, the building facade of the investigate site to which the project refers to, creating a real aspect of the studied area).

8.2 Image

This tool (Figure 8.6) lets the user import an image from an image file in common formats such as bmp, jpg, png, etc. A new node will be added at the tree menu (one new node for each created object).

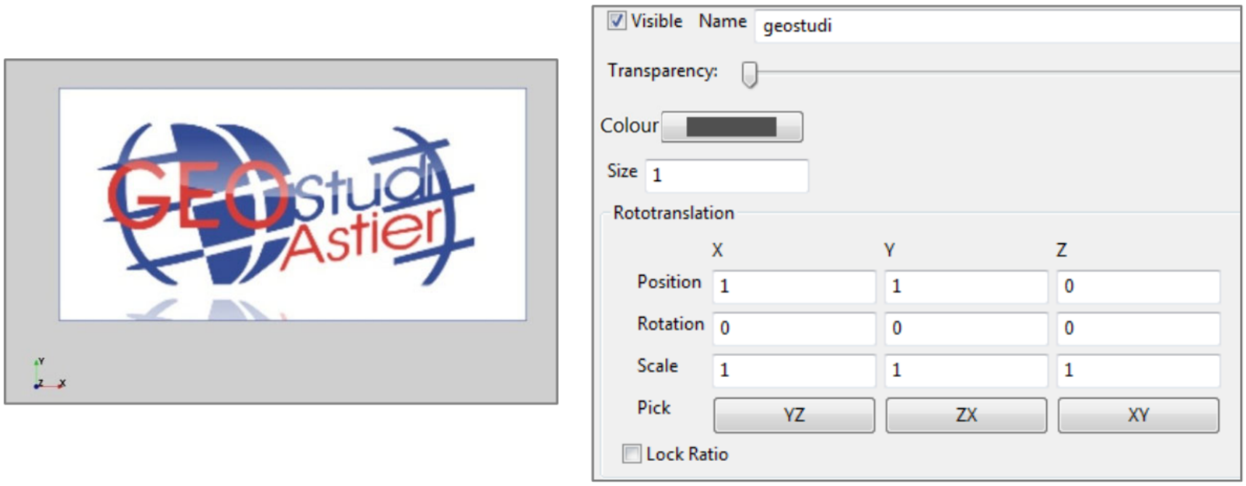


Figure 8.6: Detail of sub-node “Data” panel, property of “image”

Through the box “*Name*” it is possible to give a name to the object (“geostudi” in this example). The transparency of the image is editable through the “*Transparency*” slider. It is possible to roto-translate the image through the appropriate table (setting the Position, the Rotation and the Scale of the displayed object) or manually through the *Pick* button (see Rototranslation).

8.3 Geographic Map Image

Warning: Feature not available in the basic license; included in the “Graphics pack” add-on.

This tool let the user to plot a map in the 3D scene.

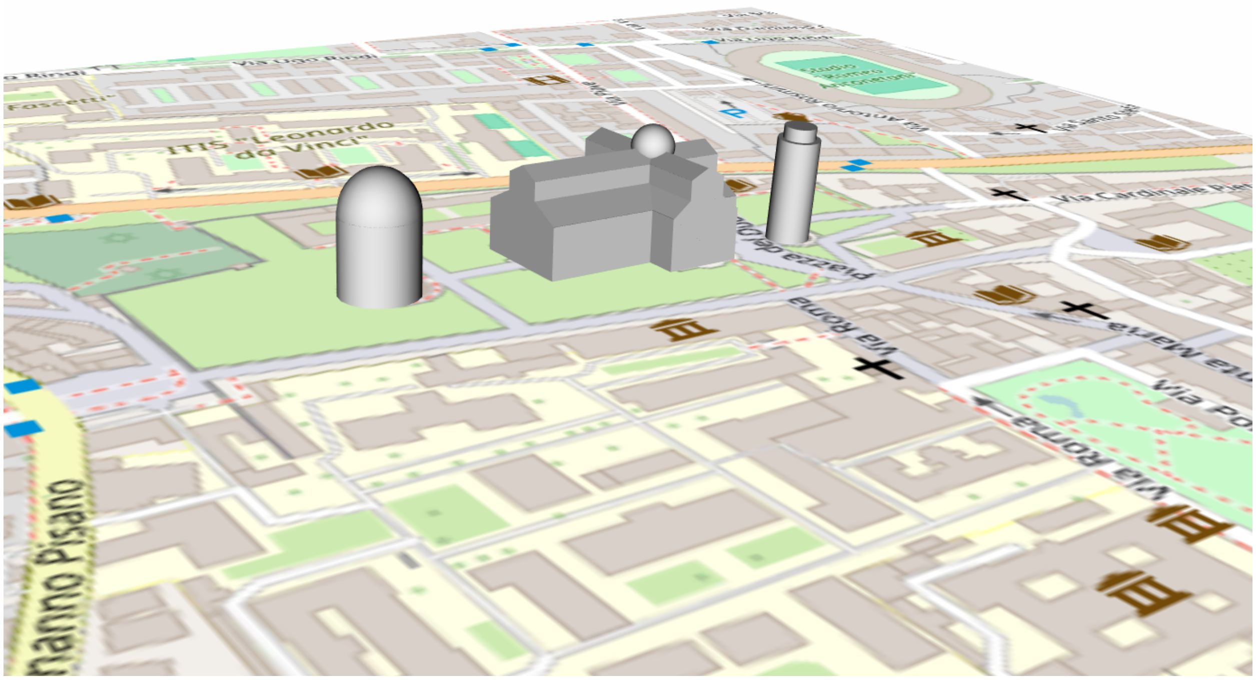


Figure 8.7: Geographic Map Image example

A map downloader is opened to select the right region to be plotted.

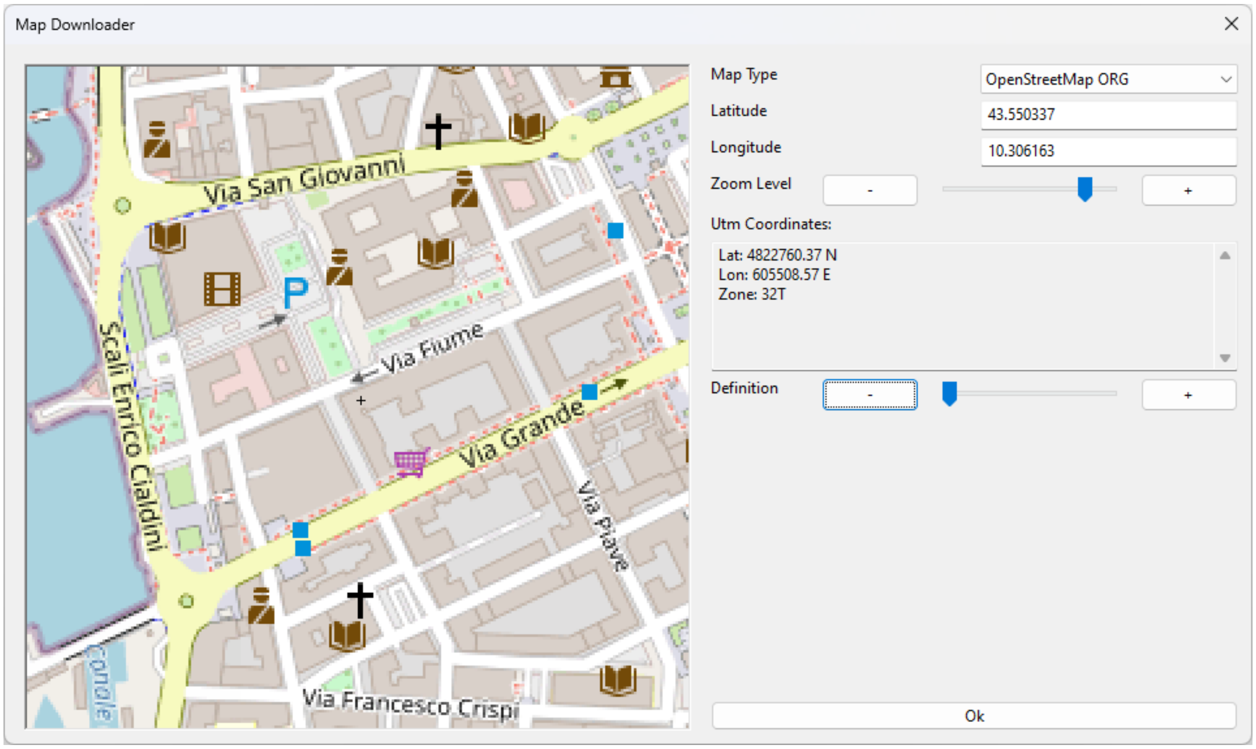


Figure 8.8: Map downloader

It is possible to select the “*Map type*” to get different type of images from specific sources. See the text file “*Map.txt*” in the installation folder.

The “*Latitude*” and “*Longitude*” can be set manually editing the displayed numbers, or, more easily, it can be translated the map with the left mouse button. The little black cross plot at the centre of the map is the point provided.

The “*Zoom Level*” can be set in a wide range, moving the slider, or using the mouse wheel on the image.

In real time the “*Latitude*” and “*Longitude*” (that are in degrees) are also converted in UTM metric coordinates and showed in the window.

Increasing the “*Definition*” it will be possible to get larger images. Note that the default map tile is 256x256 pixel, and at each increment of the “*Definition*” the image map will be one tile larger than before.

Pressing the “*Ok*” button it will ask to save the map as an image on the computer. The name used it will be the name of the new node created in the tree.

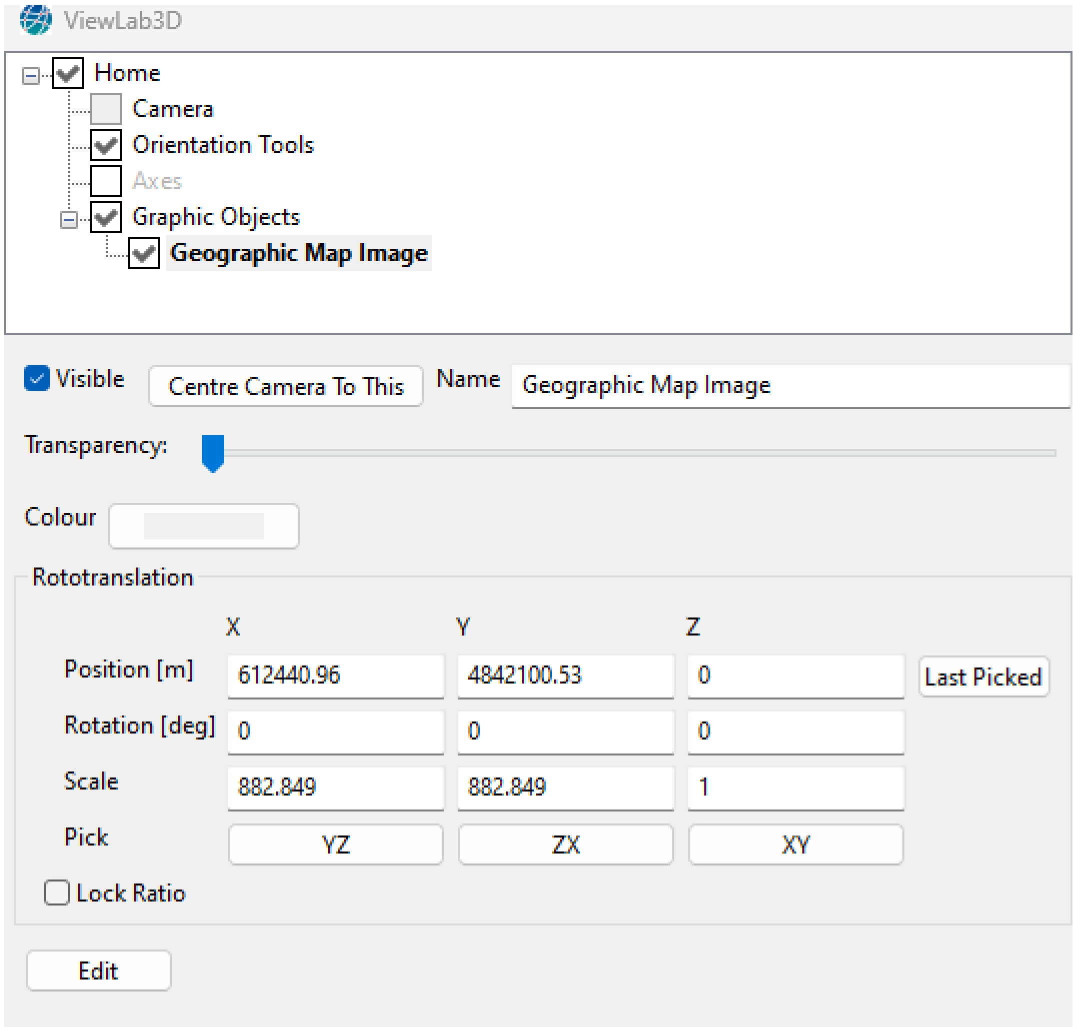


Figure 8.9: Geographic Map Image properties

The main parameters are similar to the ones that can be also found with the tool *Image*.

It will always be possible to modify the map pressing the "*Edit*" button to go back to the map downloader.

8.4 DXF

It is possible to import a CAD file (*.DXF format) selecting the appropriate button in the *Insert Object* menu. The CAD file is the only object which is automatically located in its real coordinates and not in the origin. The dxf format is very complex and constantly improving, so it possible to import just the basic elements as points, lines, poly-lines, arcs, circles, and text (and blocks made from these items); more complex objects are not imported nor represented. However, this tool is very powerful and can be used in many situations.

Figure 8.10: Example of dxf; in red, electrodes

8.5 Shape

From here it is possible to insert several 3D object in the scene, see the following options.

8.5.1 Label

It is possible to insert some labels as words, text or numbers in the scene (Figure 8.11).

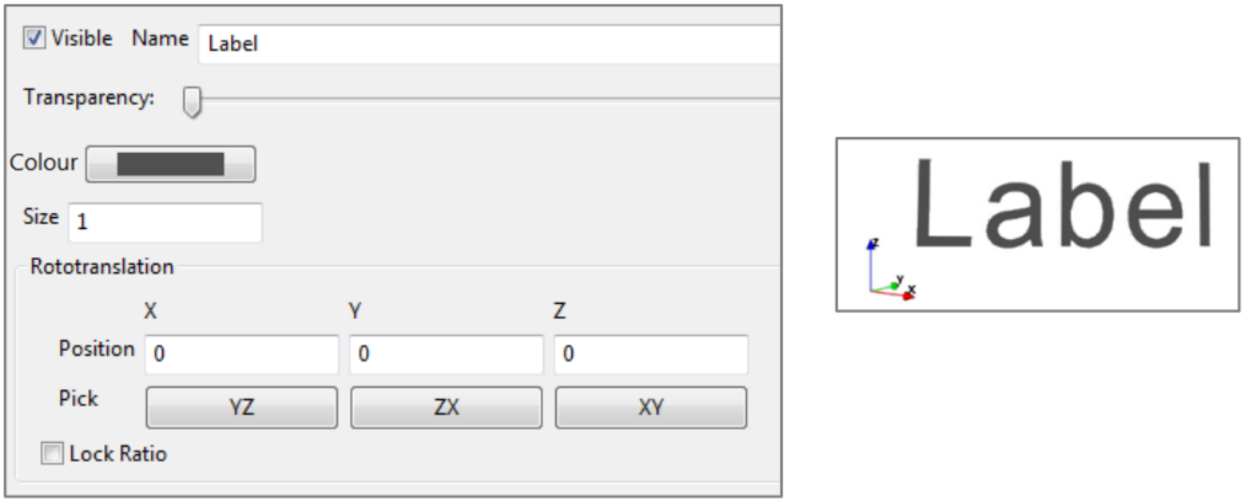


Figure 8.11: Detail of sub-node "Graphic Object" panel, property of "Label"

As for the objects explained before, it is possible to set the *Transparency* of the object, the *Colour*, the *Size*, the *Position* in X, Y and Z, to control the visibility of the label, and to give it a *Name* ("Label" in the example) by writing it in the appropriate space. It is also possible to rototranslate the label manually through the *Pick* button, see above.

8.5.2 Plane

It is possible to insert a plane in the scene through the appropriate button in the *Insert Object* menu (Figure 8.12).

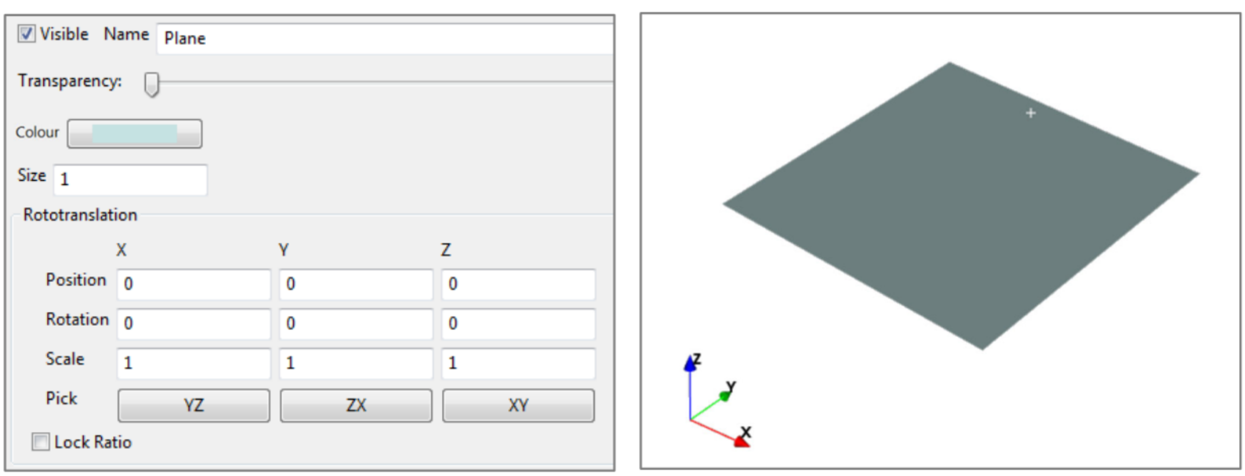


Figure 8.12: Plane Panel

As for all the objects explained before, it is possible to set the *Transparency*, *Colour*, *Size*, *Position* in X, Y and Z, Visibility, and to *Name* the object ("Plane" in the example). It is also possible to rototranslate the plane manually through the *Pick* button as explained before.

8.5.3 Cube

It is possible to insert a cube (Figure 8.13) in the scene through the proper button in the *Insert Object* menu.

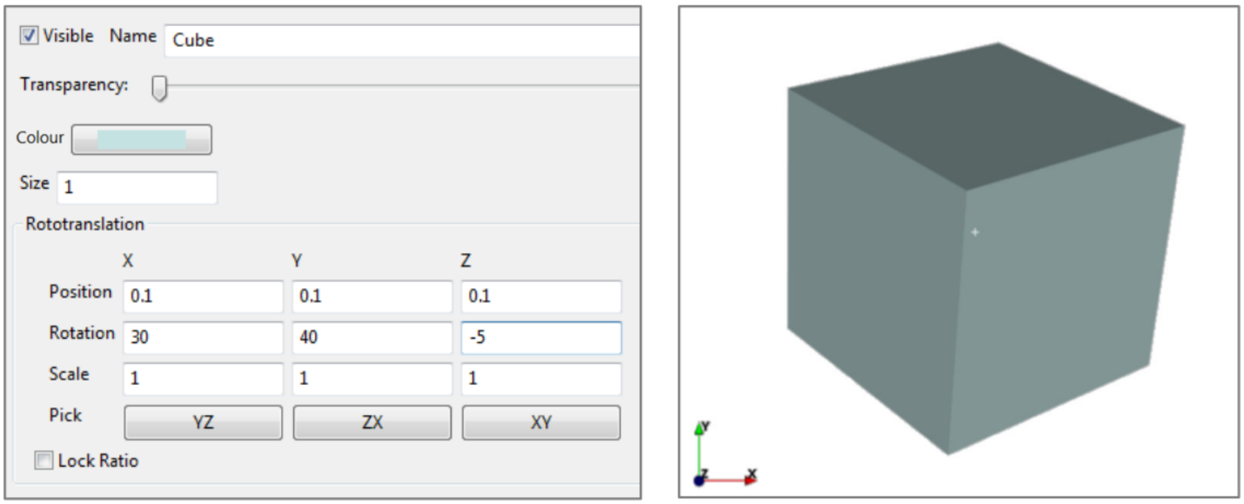


Figure 8.13: Cube panel

As for all the objects explained before, it is possible to set the *Transparency*, *Colour*, *Size*, *Position* in X, Y and Z, Visibility, and to *Name* the object ("Cube" in the example). It is also possible to rototranslate the plane manually through the *Pick* button as explained before.

8.5.4 Sphere

It is possible to insert a sphere (Figure 8.14) in the scene through the proper button in the *Insert Object* menu.

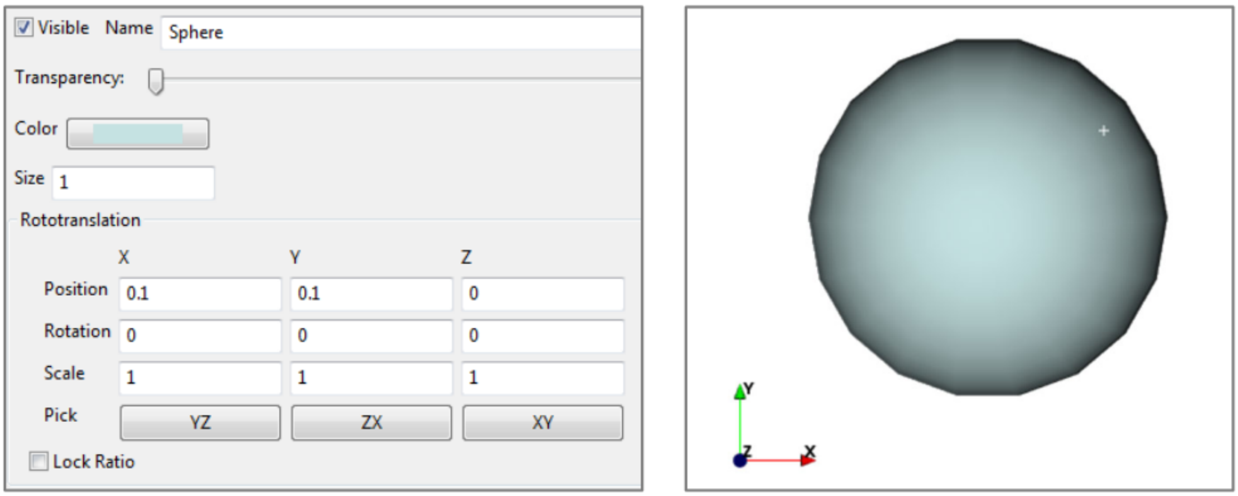


Figure 8.14: Sphere Panel

As for all the objects explained before, it is possible to set the *Transparency*, *Colour*, *Size*, *Position* in X, Y and Z, Visibility, and to *Name* the object ("Sphere" in the example). It is also possible to rototranslate the plane manually through the *Pick* button as explained before.

8.5.5 Cylinder

It is possible to insert a cylinder (Figure 8.15) in the scene through the proper button in the *Insert Object* menu.

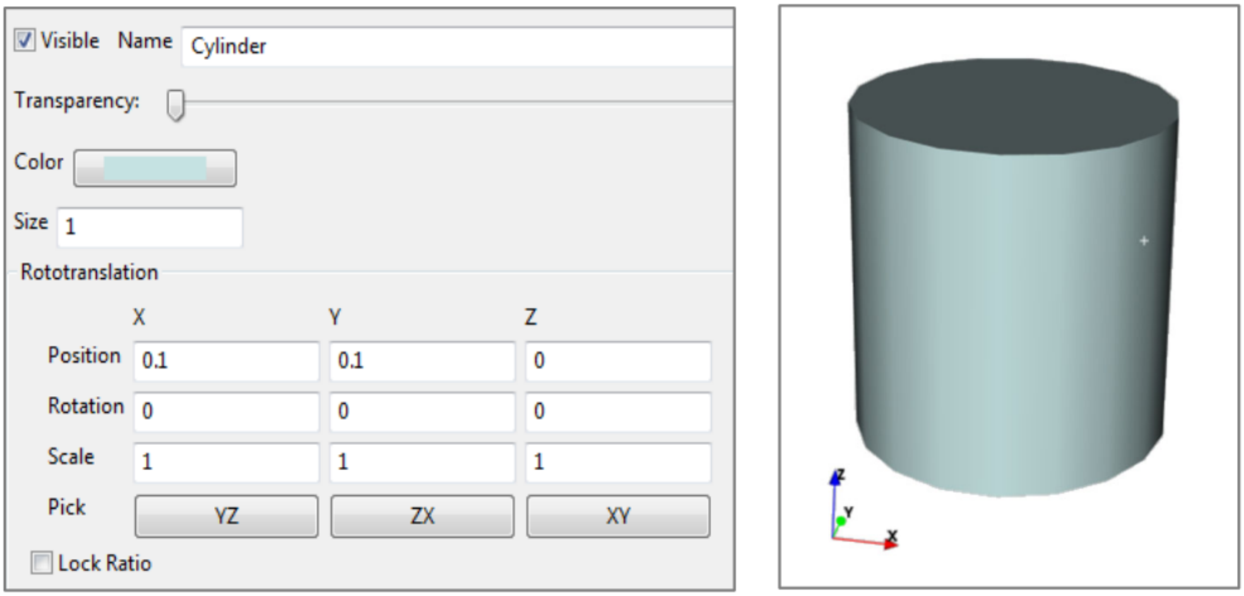


Figure 8.15: Cylinder Panel

As for all the objects explained before, it is possible to set the *Transparency*, *Colour*, *Size*, *Position* in X, Y and Z, Visibility, and to *Name* the object ("Cylinder" in the example). It is also possible to rototranslate the plane manually through the *Pick* button as explained before.

8.5.6 Cone

It is possible to insert a cone (Figure 8.16) in the scene through the appropriate button in the *Insert Object* menu.

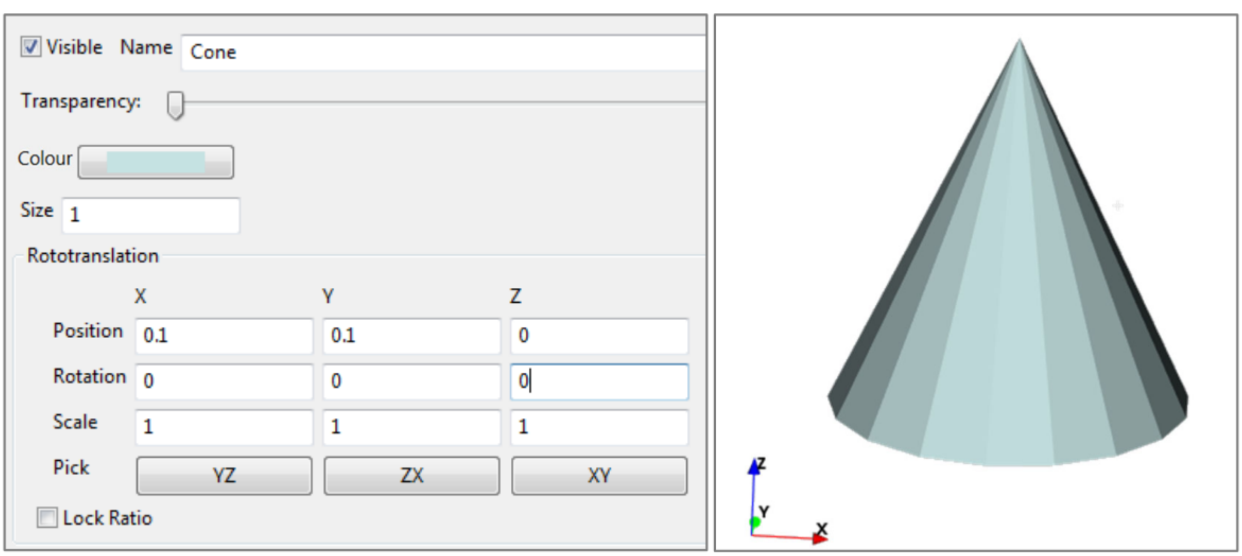


Figure 8.16: Cone Panel

As for all the objects explained before, it is possible to set the *Transparency*, *Colour*, *Size*, *Position* in X, Y and Z, Visibility, and to *Name* the object ("Cone" in the example). It is also possible to rototranslate the plane manually through the *Pick* button as explained before.

8.5.7 Pyramid

It is possible to insert a cone (Figure 8.17) in the scene through the appropriate button in the *Insert Object* menu.

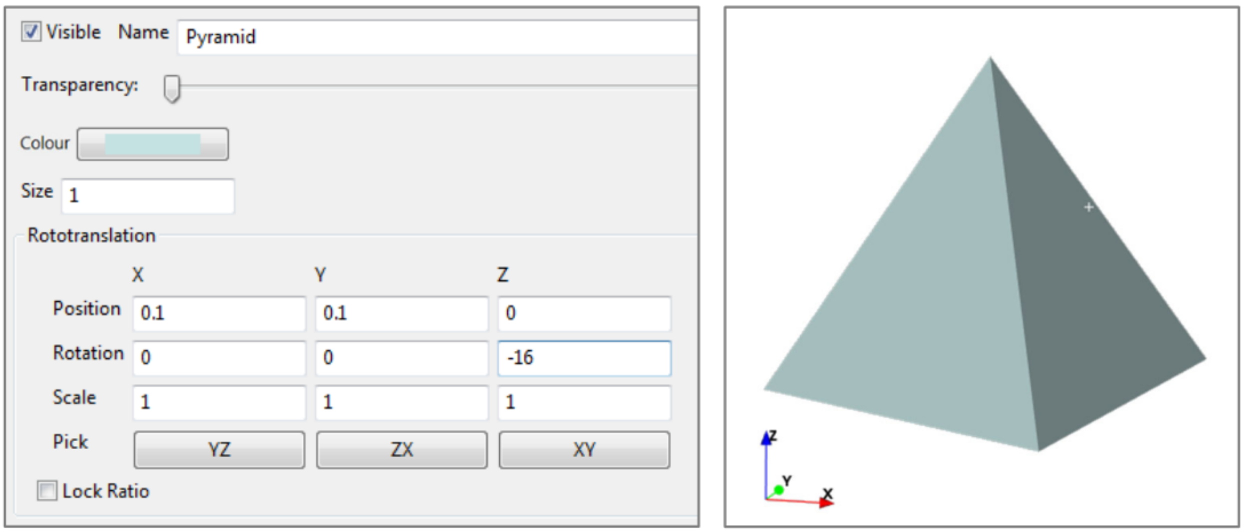


Figure 8.17: Pyramid panel

As for all the objects explained before, it is possible to set the *Transparency*, *Colour*, *Size*, *Position* in X, Y and Z, Visibility, and to *Name* the object ("Pyramid" in the example). It is also possible to rototranslate the plane manually through the *Pick* button as explained before.

8.6 Grid

It is possible to insert a grid (Figure 8.18) in the scene; it is useful to have a reference for the objects, similar to the "Axes".

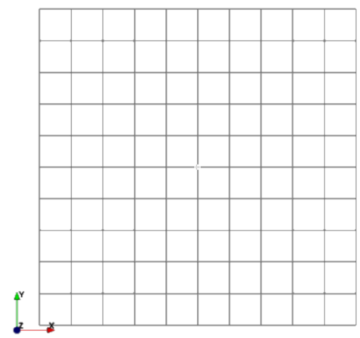


Figure 8.18: Grid Panel

In addition to the usual setting of *Visibility*, *Transparency*, *Colour*, and *Rototranslation* there is the option to set the *Orientation* of the grid and the *Number of cell* to insert in two dimensions (inside the dashed red line in Figure 8.19).

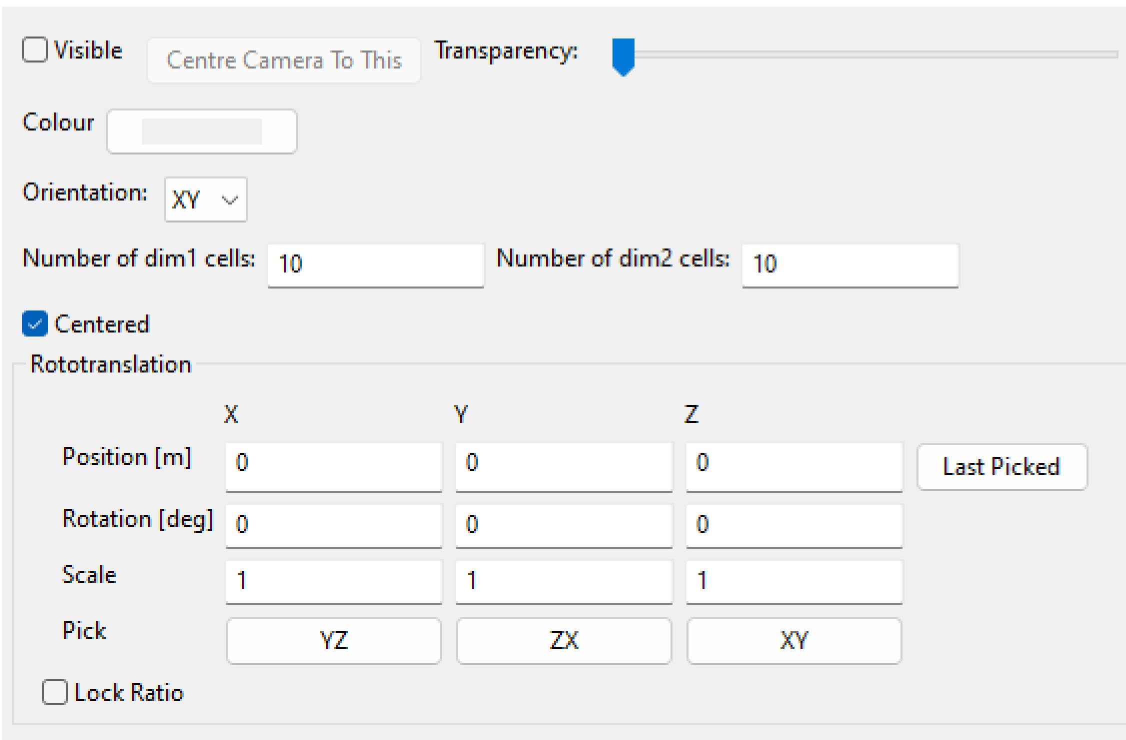


Figure 8.19: Grid panel

By selecting one of the 3 options available in "*Orientation*" (3 primary plans "XY", "XZ" or "YZ") it is possible to choose the plane on which to visualize the grid, as shown in Figure 8.20.

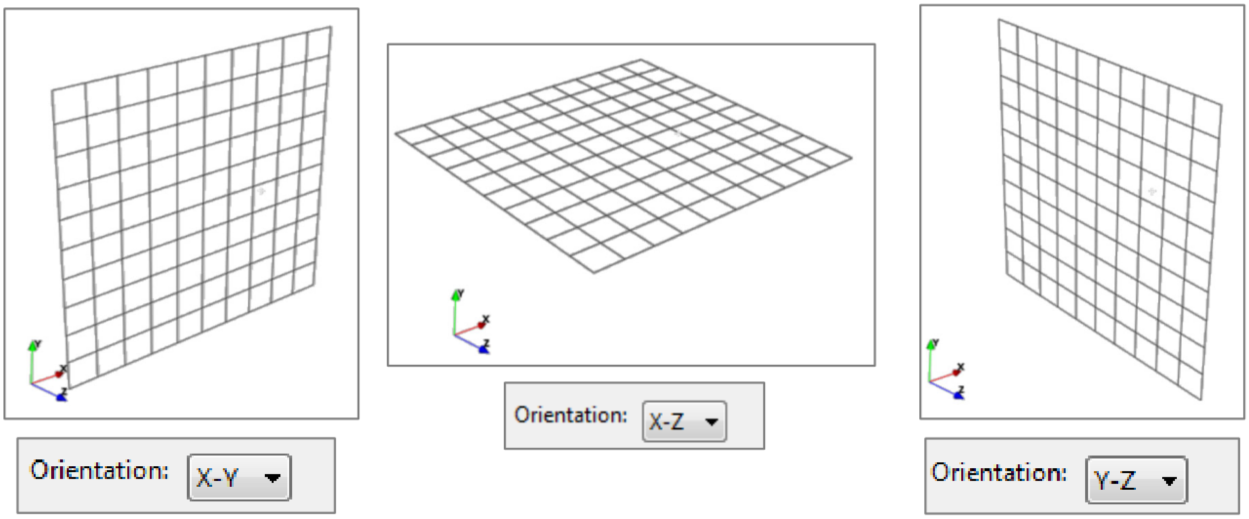


Figure 8.20: Different orientation of the grid in the 3D space

To set the size of the grid use the slider or insert the number of grid boxes (10 by default) manually between the minimum dimension of 1 box to the maximum dimension of 100 boxes. The relative minimum and maximum value for the grid box number selection can be set (in orange in Figure 8.21) by writing the corresponding number in the dedicated boxes. This way the user can change the limiting values of the slider from their defaults 1 and 100 to the new selected values.

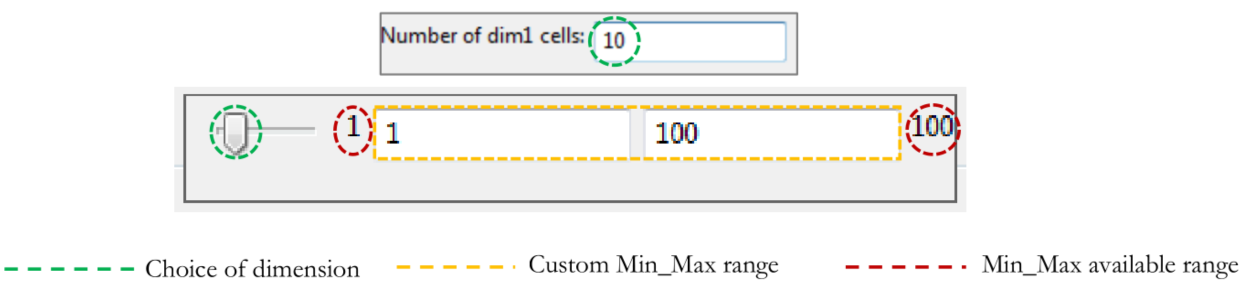


Figure 8.21: Changing maximum and minimum values

For example, if the minimum size should be 15 and the maximum 60, writing this numbers in the boxes the slider will show 15 as the minimum value when it is totally on the left, 60 as maximum value when it is totally on the right, and any intermediate value of the sliding bar (Figure 8.22).

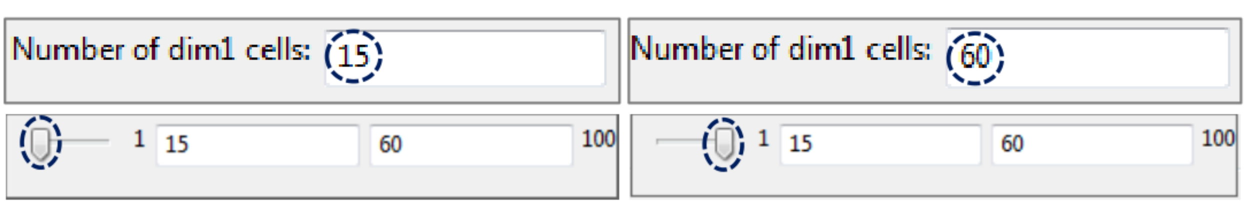


Figure 8.22: Example of setting of minimum and maximum number of cells

The two dimensions are separately editable (Number of dim1 and dim2 cells in Figure 8.23).

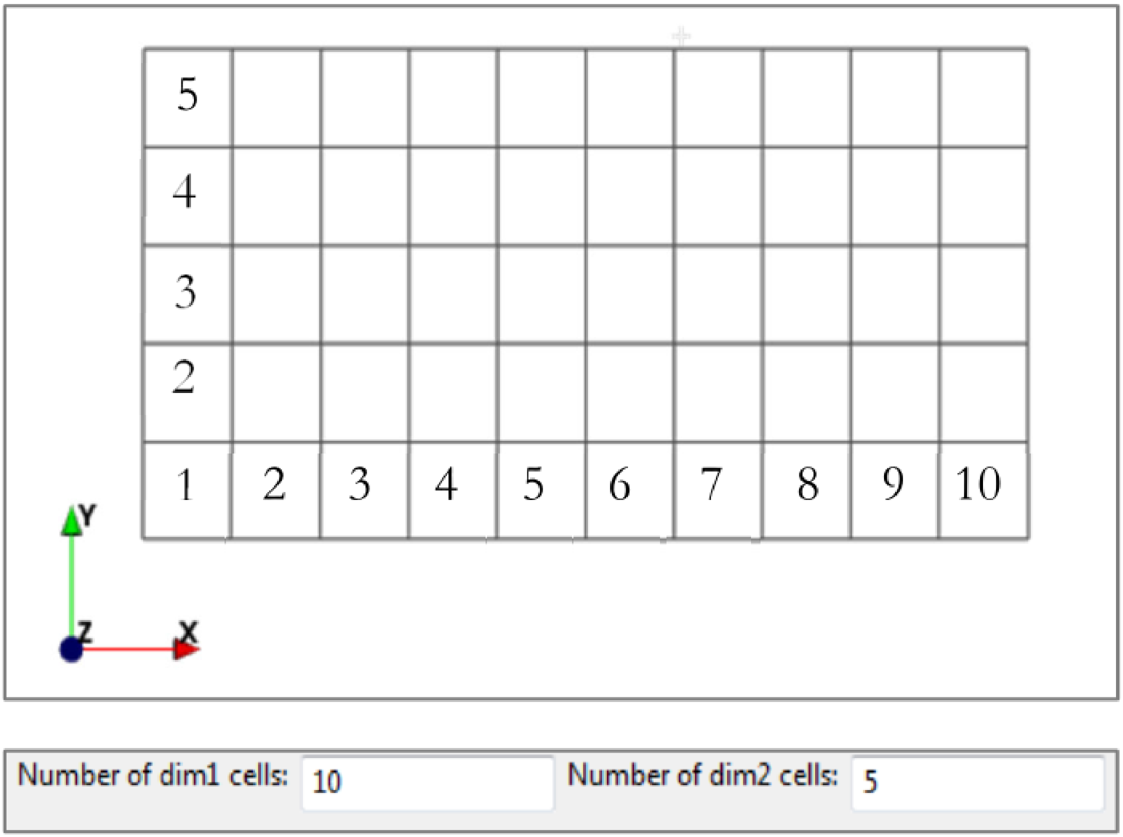


Figure 8.23: Example of number of cells setting in the two dimensions

8.7 Axes

This node has exactly the same functionalities of the global node Axes. The main benefits of using this option rather than the global one are:

- it is possible to have multiple axes to the same 3D scene without the need to edit a single node more times to achieve a similar result
- it is here possible to save/load all node settings to/from file
- inserting axes into an inner (not global) *Graphic Object* node (like the one into an ERT project node) helps to have automatically saved its settings together with the project saving step.

8.8 Core Sample

Warning: Feature not available in the basic license; included in the "Graphics pack" add-on.

It is possible to insert a core sample in the scene as shown in Figure 8.24. A sequence of vertical cylinders are drawn to show the core in the 3D scene.

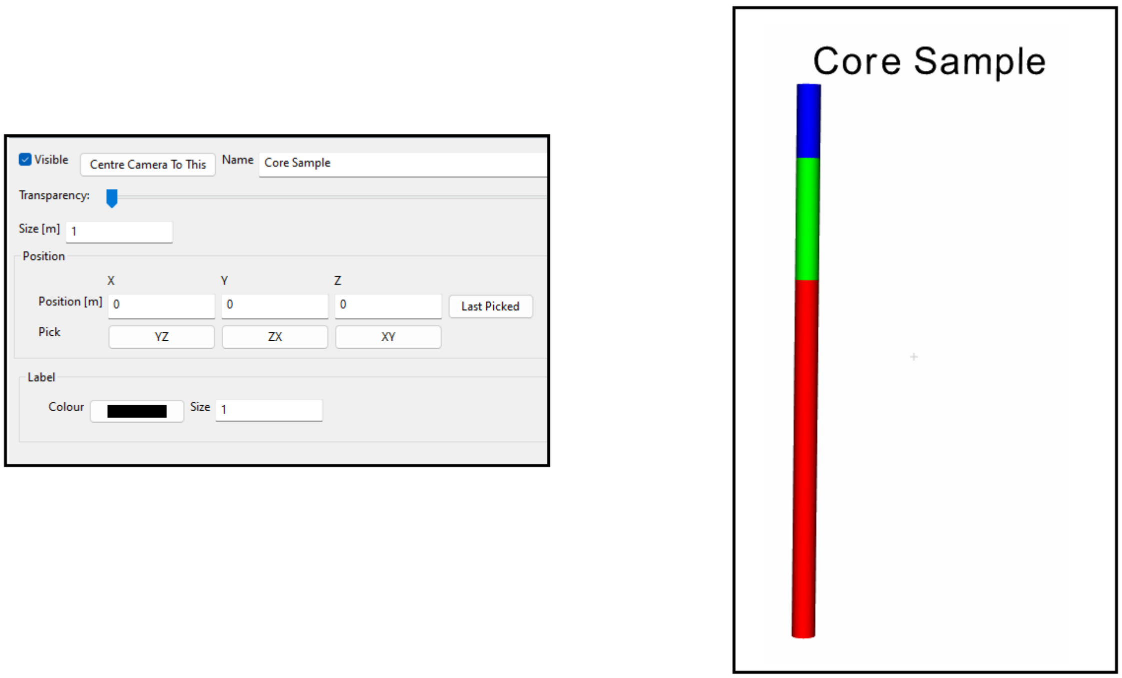


Figure 8.24: Core Sample panel

The properties available are, as usual, *Visibility*, *Name*, *Transparency*. It is also possible to set the *Position* of the top of the core, and the diameter of the cylinders (see *Size*). The given *Name* will be also plot in the 3D scene near the top of the core, the font *Colour* and *Size* can be set (see *Label* group).

To customize the *Core Sample* it is possible to use some tools available right clicking on the node.

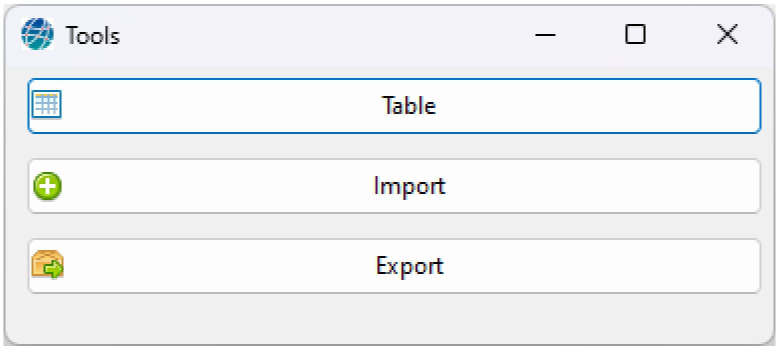


Figure 8.25: Core Sample edit tools

From Figure 8.25 it can be noted that one option is to open and edit a table to set the properties of each layer. The tables can be used to set a thickness and a colour to each layer. It is also possible to set names, that are globally managed by the software, so they can be shared between multiple nodes to easily link the same colour to the same type of material.

The buttons at the bottom of the panel are for *Append* a new item at the end of the table, or to *Delete* all the currently selected rows.

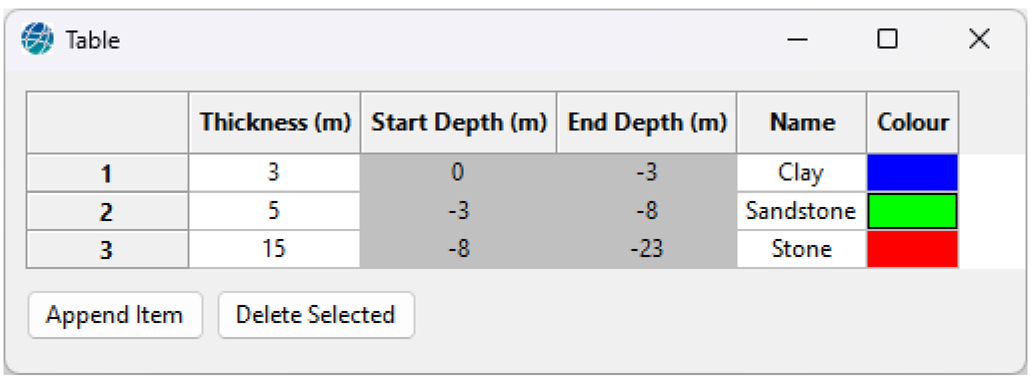


Figure 8.26: Core Sample Table

From Figure 8.25 it can be note that other options to edit the *Core Sample* are an *Import* / *Export* from/to text files. The format of the text file is like the example below, where the information are listed in three columns, that describes “Thickness”, “Name” and “Colour” for each layer.

```
3   "Clay"         "#0000FF"
5   "Sandstone"    "#00FF00"
15  "Stone"        "#FF0000"
```

8.9 Point Cloud

Warning: Feature not available in the basic license; included in the "Graphics pack" add-on.

It is possible to insert in the 3D scene a cloud of points that can represent many type of information, like a DTM (Digital Terrain Model) data for the field surface topography description, or a LaserScan dataset for a 3D view of the surrounding scenario.

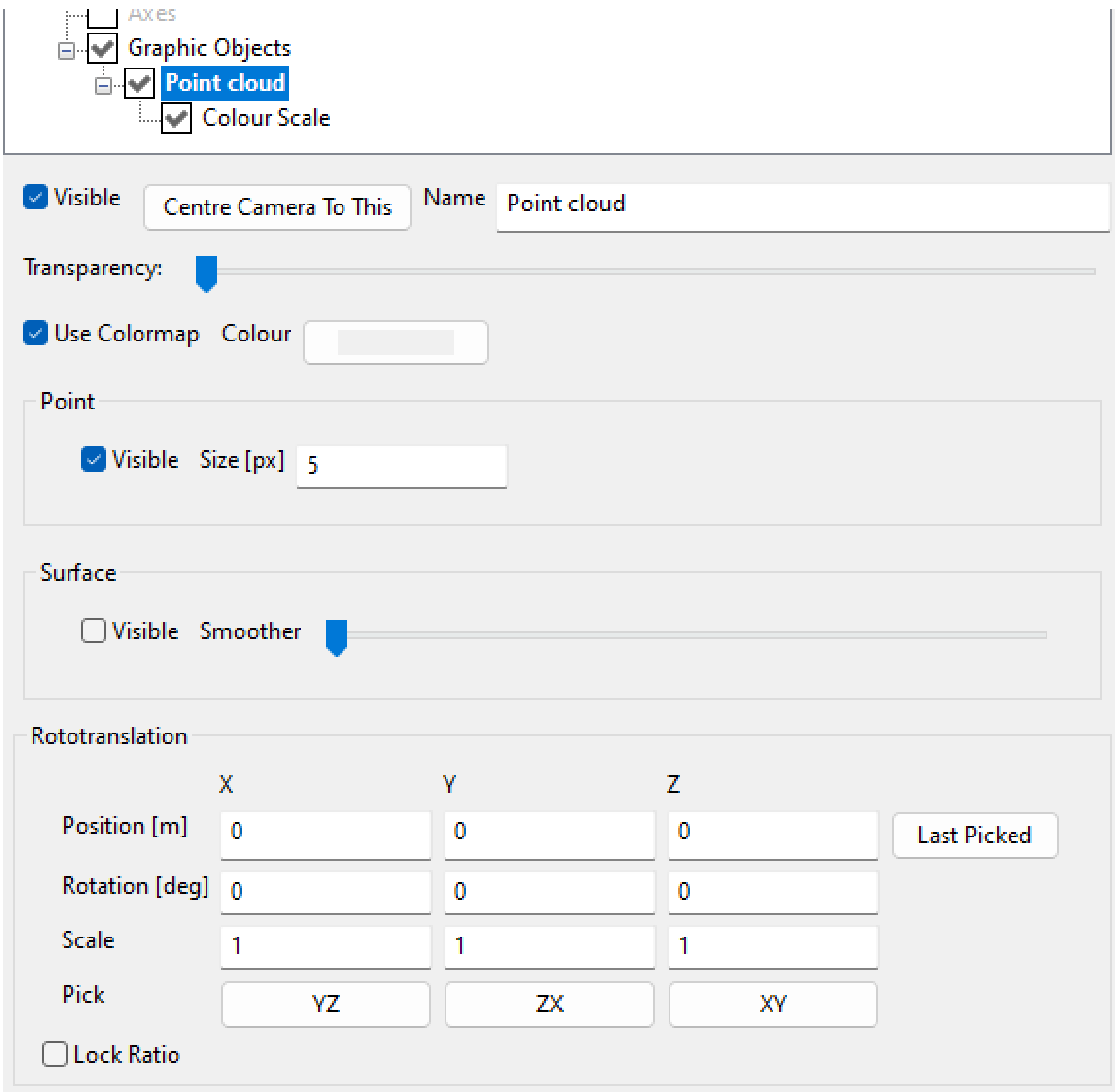


Figure 8.27: Point Cloud panel

The properties available are, as usual, *Visibility*, *Name*, *Transparency*. It is possible to set the point *Colour* to an homogeneous one, or, if available, to use the colour provided provided for each point (see *Use Colormap* flag). It is possible to set the *Size* (in pixel) of each point. In addition to (or as an alternative to) the point visualization it is also possible to interpolate all the provided points with a surface (that can also be smoothed, if desired). It is possible to rototranslate the *Point Cloud* through the appropriate table (setting the Position, the Rotation and the Scale of the displayed object) or manually through the *Pick* button (see Rototranslation).

When trying to create this kind of node it is asked to select a file from the computer and to select the type of import, as shown in Figure 8.28. The selected file needs to be in text format, usually in numeric format only, with many lines as the number of points to be loaded, and the correct number of columns according to the type of import selected.

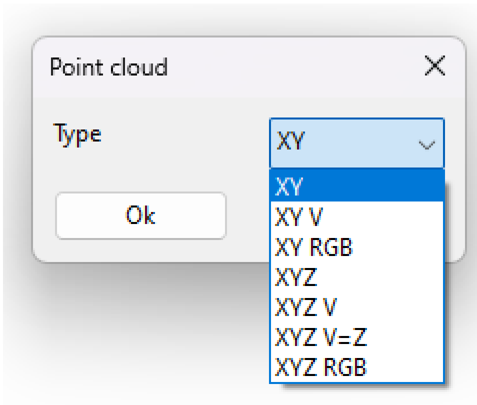


Figure 8.28: Type of import

- **XY** for flat point list (Z is assumed equal to 0). The number of columns required in the text file is 2. An homogeneous colour will be applied to all the points.
- **XV (V)** for flat point list (Z is assumed equal to 0). It will be also managed an additional value (V) for each point, that will be related to the point colour. The number of columns required in the text file is 3.
- **XY (RGB)** for flat point list (Z is assumed equal to 0). It will be also managed three additional values for each point, with RGB (red-green-blue) format, that will explicitly describe the point colour. The number of columns required in the text file is 5.
- **XYZ** for 3D point list. The number of columns required in the text file is 3. An homogeneous colour will be applied to all the points.
- **XYZ V** for 3D point list. It will be also managed an additional value (V) for each point, that will be related to the point colour. The number of columns required in the text file is 4.
- **XYZ V=Z** for 3D point list. It will be also managed an additional value (V) for each point, that will be related to the point colour. That V values will be an implicit copy of the Z values. The number of columns required in the text file is 4.
- **XYZ RGB** for 3D point list. It will be also managed three additional values for each point, with RGB (red-green-blue) format, that will explicitly describe the point colour. The number of columns required in the text file is 6.

Note that when the additional scalar value (V) is provided together the point position is then created also the sub-node Colour Scale to customize how the value needs to be rendered as colour.

In Figure 8.29 some rendering examples starting from a similar dataset:

- **A** shows an **XY V** flat Point Cloud, with colours
- **B** shows an **XYZ** 3D Point Cloud, without colours
- **C** shows an **XYZ V** 3D Point Cloud, with colours
- **D** shows an **XYZ V** 3D Point Cloud, with colours and interpolated into a surface

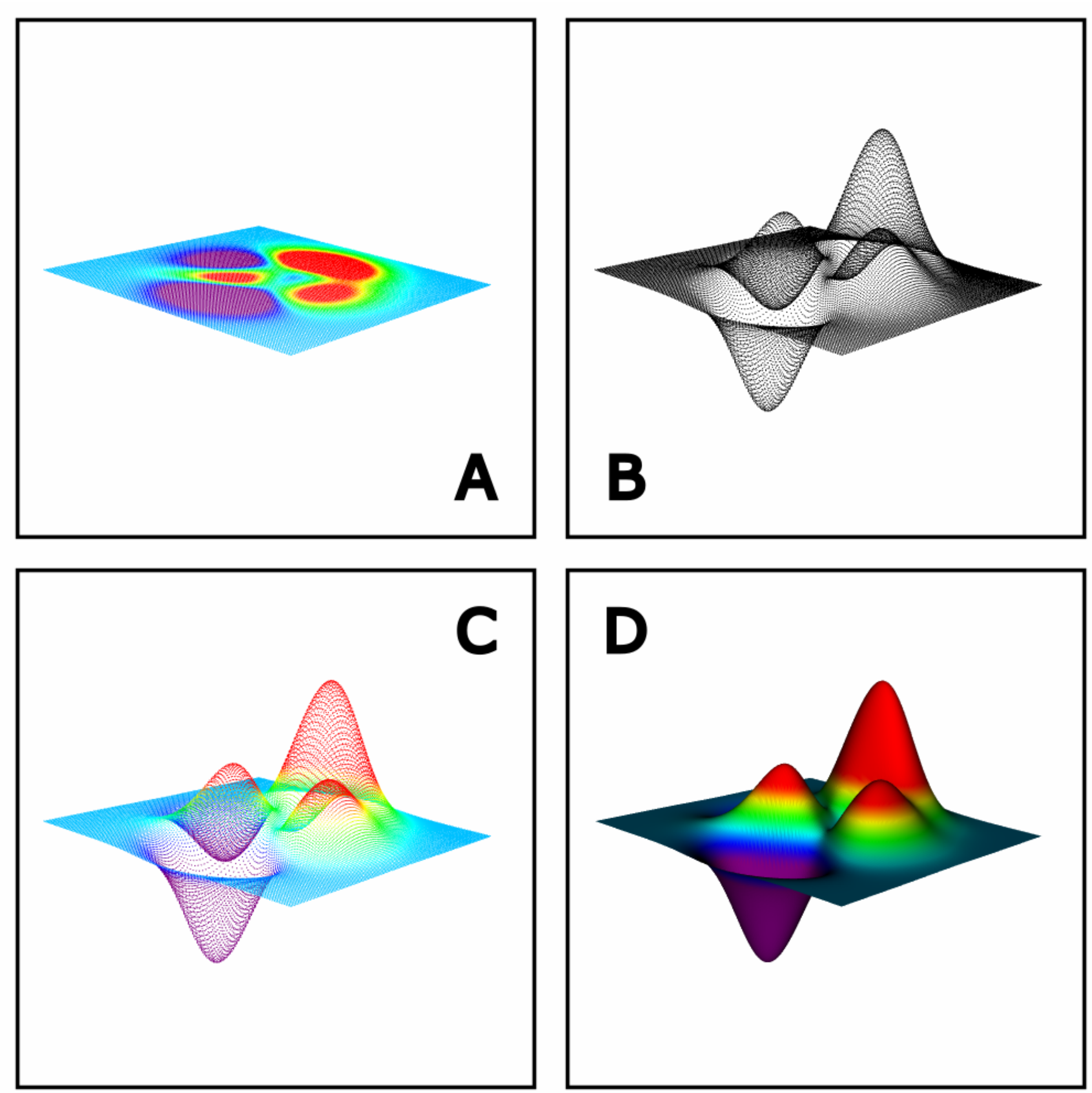


Figure 8.29: Point Cloud examples

CHAPTER

NINE

ERT DATA

Some main functionalities related the *ERT Data* projects can be found in the window that opens automatically when the program is launched. Clicking with the right mouse button on the *Home* node and then on the action tool, the window can be opened again (Figure 9.1).

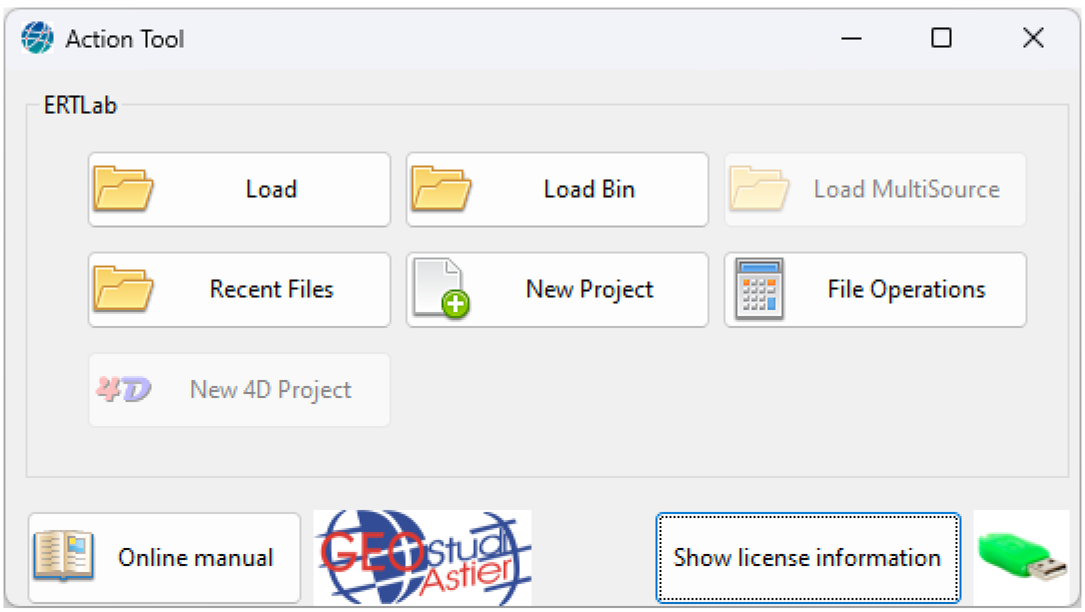


Figure 9.1: Action Tool panel

According to the type of license activated (see the green USB protection dongle) the number of buttons shown and activated can change.

At the bottom of that window it can be found a small image that tells if the USB dongle is insert (the image is coloured) or not (the image is greyed). If it is needed to check if the USB dongle status it can be pressed the button *Show license information*, if there is any problem the message shown should be similar to Figure 9.2.

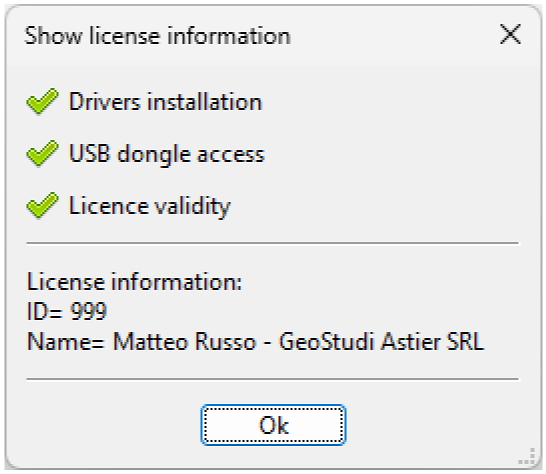


Figure 9.2: Show license information

At the bottom of that window there is also a button to open the *Online manual*, in case of needs.

The main buttons available in the tool window are:

- **Load**: opens a *.DATA file, a project that has already been created and saved.
- **Recent Files**: opens a recently opened project.
- **Load Bin**: loads a *.BIN file, a field data file collected with IRIS Syscal instruments. See the Conversion table for a smarter file import.
- **Load Multisource**: this is an additional package to process datasets acquired using the multi-source methodology (measurements are made while multiple dipoles transmit simultaneously). It loads a *.wDat file, a field data file collected with MTP-IRIS Multisource instruments.
- **New Project**: opens an empty project, it adds the Data node to the tree but without loading any data files.
- **File Operation**: loads two projects and makes operations between them; it is a useful tool for time lapse evaluation (to compare the resistivity response before and after an event).
- **New 4D Project**: this is an additional package to run time-lapse inversions using the differences approach.

9.1 File Operation

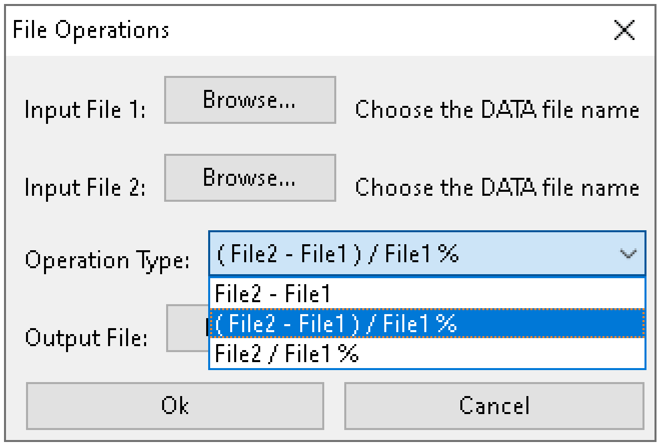


Figure 9.3: File Operation

It is needed to specify the input DATA file name 1 and 2, where it will be loaded the 3D models (usually obtained from an inversion, see section Inversion).

It is also needed to specify the type of operation that needs to be used to combine the input meshes. Usually this tool is used to compute a percentage variation, so the option shown in the Figure 9.3 it is then used.

An output file is then saved (in the location specified), and then it is automatically opened to view the results immediately.

It is quite common to set the colormap shown in Figure 9.4 , being also careful to put identical extremes so that the zero value is associated with the white colour.

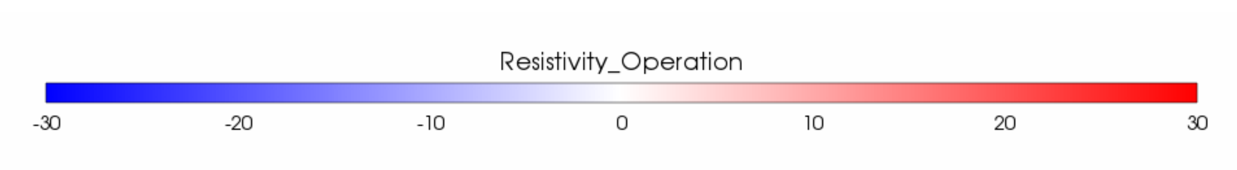


Figure 9.4: Colormap example

9.2 Timelapse 4D Project

Warning: Feature not available in the basic license; included in the "4D" add-on.

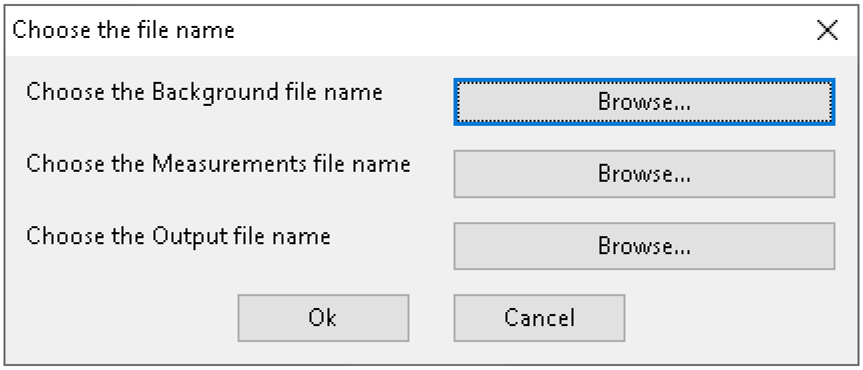


Figure 9.5: New 4D Project

To create a 4D project it is needed to specify two input DATA filenames. These derive from the saving of measurements that took place at different times. The purpose of this tool is to help achieve a stable joint inversion, to evaluate the differences in the model. The requirement is to use electrodes in exactly the same position, and the sequence of measurements must be identical (or as similar as possible). The first file is defined "Background", it is usually temporally prior to the other, that is defined "Measurements".

The "Background" DATA file name must contain:

- Electrodes and field measurements
- A Model, obtained from the 3D inversion of the field data
- The inversion parameters used to get the Inverted
- The calculated data obtained from the forward model computed on the Model and the measurement sequence given

Note that all these information are always available automatically after an usual processing from the field data to the inverted model.

The "Measurements" DATA file name must contain:

- The same electrodes and field sequence measurement.

Any mesh and inverted model needs to be provided in the "Measurements" DATA, also any other parameter need to be specified (for example parameters to compute the mesh, or to perform the inversion).

As usual, raw data cannot be used to successfully complete an inversion process, but a filter operation must be performed. Since this step will most likely bring, for each measurement, a different number of surviving quadrupoles, then the program will autonomously identify the subset common to the two files in order to correctly set the 4D project. In order to carry out any analysis on this step, a copy of "Background" and "Measurements" DATA file it is then saved, storing only the common quadrupoles, it is added the postfix "_Updated" to the original name.

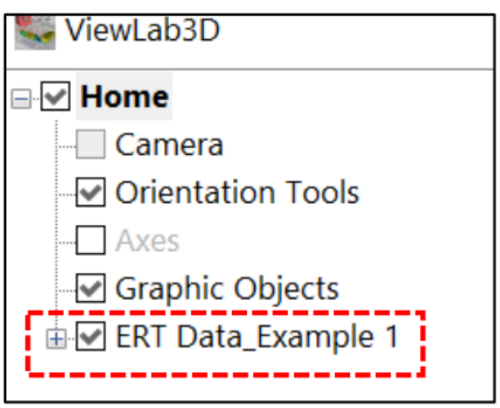
The output 4D project is then saved in the location specified, then it is automatically opened by ERTLab *Studio*. It is then only needed to run the inversion process, without editing the parameters given (taken automatically from the "Background" DATA project). At the end, the inverted 3D model will be available for viewing, which can also be compared with the reference model ("Background") by percentage variation (see section *File Operation*).

Obviously, a similar result can also be obtained by comparing separate classic inversions of the two available data, but since the two inversions will be independent in this way, then it is known that many more artifacts will appear from the comparison of the results. The 4D method helps to have more homogeneous inversions, and to considerably mitigate the presence of such artifacts.

9.3 Main project node

An ERT project will create in the tree the node *ERT data*. Whatever kind of file is opened, a lot of nodes are added to the tree, as child of *ERT data*.

In Figure 9.6 the new node created in the tree, and its property panel.



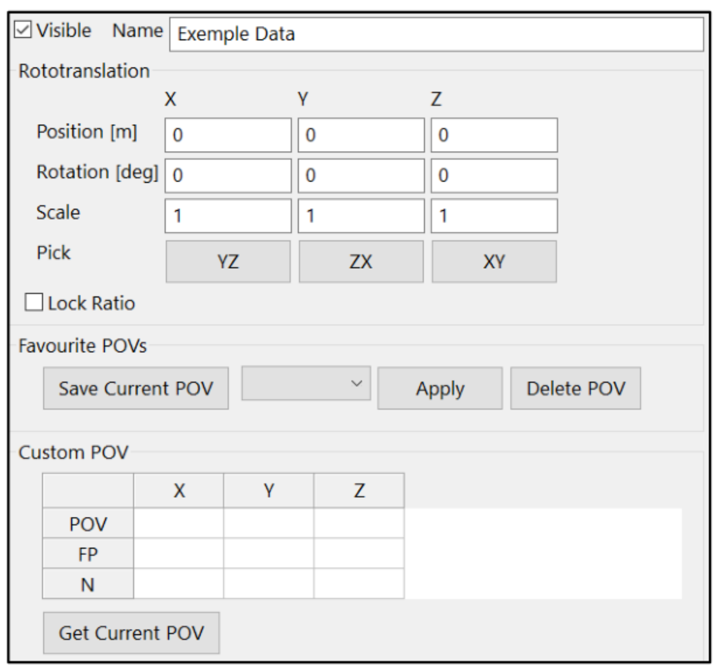


Figure 9.6: ERT Data panel

Note that the *Name* of the project was renamed to *ERT data_Example 1*. It can be also set if the project needs to be *Visible* or not. With *Centre Camera to this* the user can automatically place the selected project at the centre of the scene. It is particularly useful in those cases when many projects are loaded and the selected one is not visible, because it is out the scene (by default, the scene is centred to view all the objects in the scene).

A Rototranslation can be set to the entire project. This is only a **"graphic" rototranslation**, the actual coordinates of the data file do not change. It is a useful tool to visualize the project in a coordinates reference system different from the one the data were collected in. For example, in the case that the axes are in a new coordinate system, but you can still export the data file, nothing has changed and it is still in the original coordinate system or to make some quick evaluation tests to find the correct rototranslation values to apply to the ERT data to actually rototranslate it through the proper tool (see Rototranslation).

As it is available in the node Camera, it is possible to manage some *Favourite POVs*. Note that the *Favourite POVs* will be stored automatically when the ERT project will be saved, else the *Favourite POVs* in the Camera node will be lost when the program is closed.

Clicking with the right mouse button in the ERT Data node it is possible to:

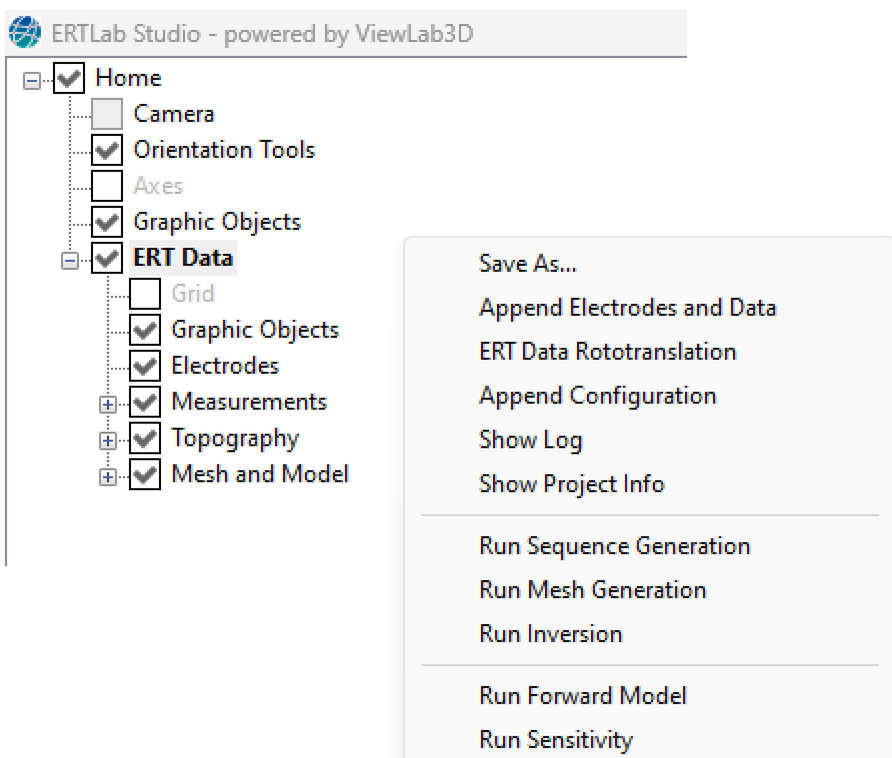


Figure 9.7: ERT Data tools

- **Save As**: through this button it is possible to save the Data-File at any time.

- **Append Electrodes and Data**: through this tool the user can append electrodes and data belonging to another data file to the current project. It is useful to merge two or more ERT lines together.

  Merging two or more lines into one project can be accomplished in two different way:

  – It is possible to load the first file with *Load/Load Bin/Load Multisource* and then add the second file through the "*Append Electrodes and Data*" tool. In this way the project maintains the name of the first loaded project (but it is possible to change it in the *Name* box later)

  – It is also possible to create an empty project with the *New Project* button and then upload all the file together through the "*Append Electrodes and Data*" tool. In this way the project maintains the default name *ERT Data* (but it is possible to change it in the *Name* box later)

- **Append Configuration**: through this option the user can append the configuration used in another previous project to the current project. This includes all the setting of the Run Mesh Generation and the Run Inversion Panels.

- **ERT Data Rototranslation**: this tool enacts an *effective rototranslation* of the dataset. It is unlike the *graphic* rototranslation and in above in this paragraph (*ERT Data panel*) (see Rototranslation). Using this tool and then exporting data loses the original reference system at the favor of a new coordinate system.

- **Show Project Info**: it opens a window with the *File Summary* (Figure 9.8). Information about the number of electrodes and measurements of the project, the dimension of the mesh and the possible presence of topography file, and Resistivity/IP/Sensitivity Models are reported. Originally, Resistivity IP and Sensitivity are "*Empty*" because no inversion has been computed yet. When a Resistivity/IP inversion is computed the proper menu items become enabled, because it is recognized the presence of an inverted Model. For the IP menu there is an intermediate state: "*Homogeneous*". It suggests that there is a unique IP value coming from the generation of the mesh (and therefore the project contains IP measurements), but the inversion has not been computed yet. In the following table all the possible cases are summarized.

	Empty	Homogeneous	Found
RESISTIVITY	There is no inverted Resistivity model in the project	—	The project contains a Resistivity model which came from an inversion process
IP	There is no inverted IP model in the project	The project contains an homogeneous IP value, but there is no inverted IP model yet	The project contains an IP model which came from an inversion process
SENSITIVITY	There is no calculated Sensitivity model in the project	—	The project contains a calculated Sensitivity model

In the example in Figure 9.8 the loaded project:

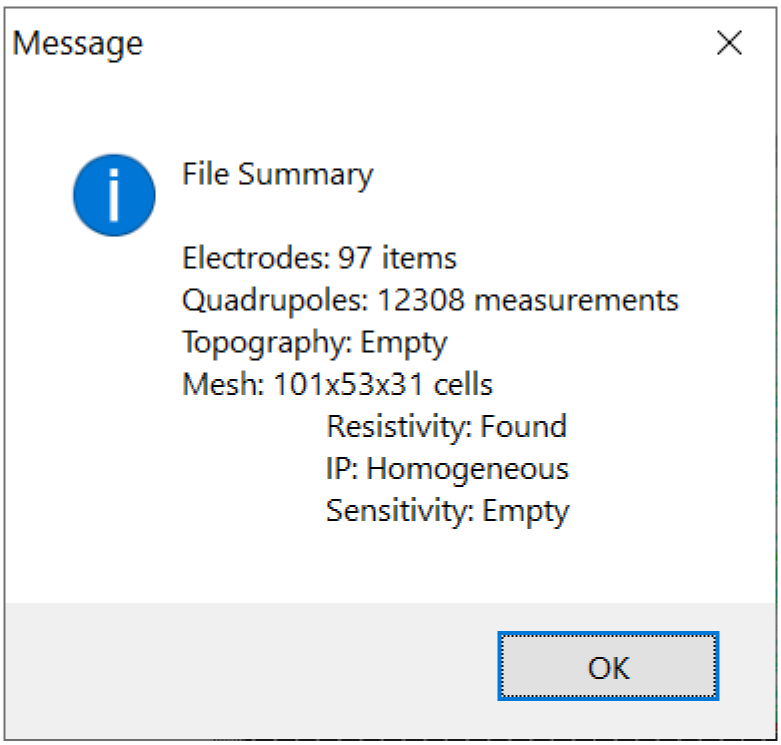


Figure 9.8: Example of Summary File message

- contains 97 *Electrodes*;
- contains 12308 *Measurements*;
- has no topography information (*Topography*: Empty);
- the *Mesh* has been calculated (accordingly the IP menu it is “Homogeneous”) and it is composed by 101 x 53 x 31 cells;

- contains a Resistivity Model, which came from an inversion already computed (*Resistivity*: Found);
- doesn't contain an *IP Model*, but a homogeneous IP value is present, which came from the mesh generation;
- doesn't contain a calculated Sensitivity Model (*Sensitivity*: Empty).

Other commands available in the *ERT Data* node are:

- **Run Sequence Generation**: see Run Sequence Generation.
- **Run Mesh Generation**: see Run Mesh Generation.
- **Run Inversion**: see Run Inversion.
- **Run Forward Model**: see Run Forward Model.
- **Run Sensitivity**: see Run Sensitivity.

This main node "ERT Data" represents the entire project and is composed of other menu items, which are described in more detail below.

9.4 Grid

This node has exactly the same functionalities of the *Grid* node available into the Graphic Objects node.

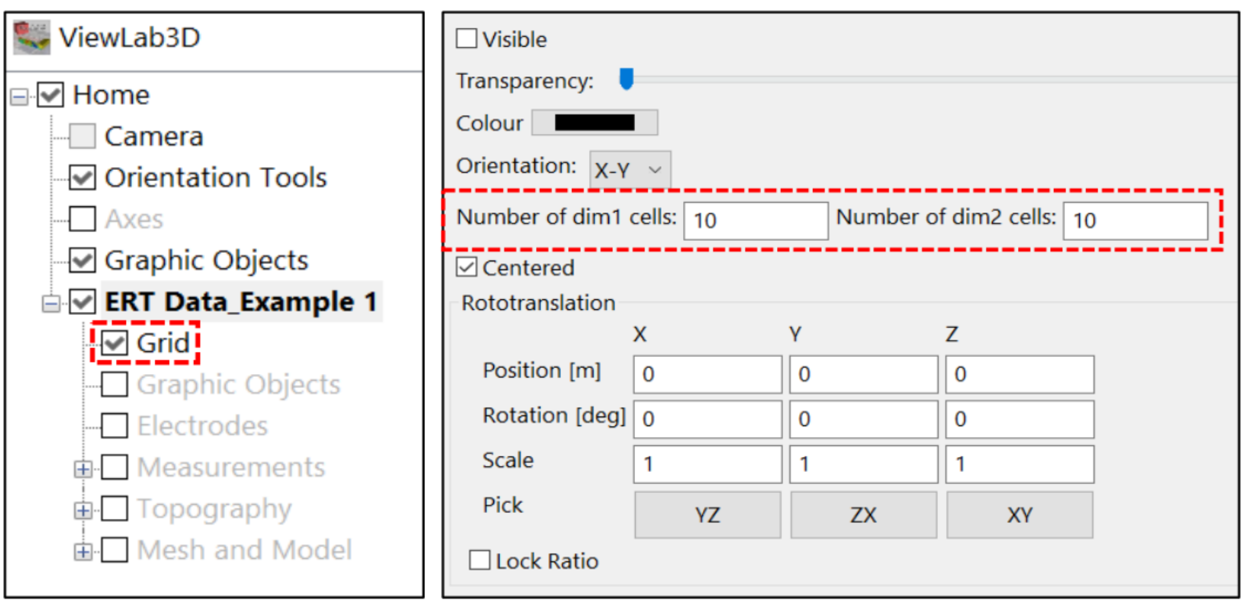


Figure 9.9: Grid Panel in ERT Data node

9.5 Graphic Object

This element is completely analogous to that described Graphic Objects , refer to it for more information.

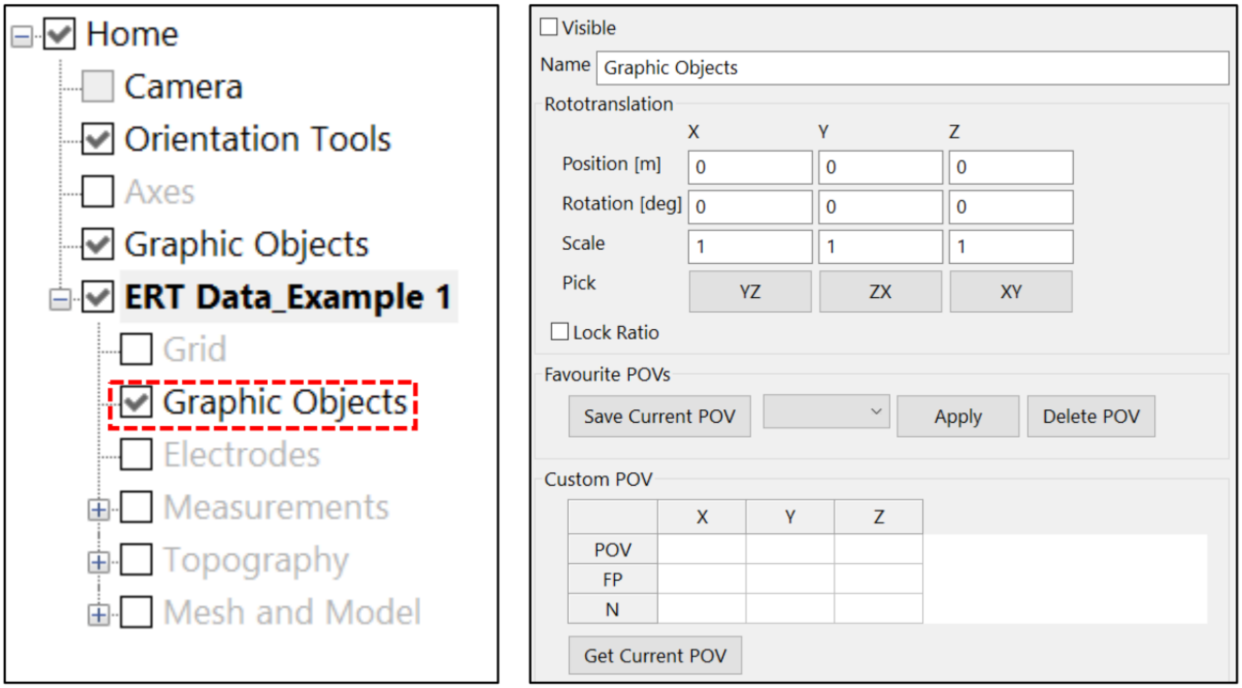


Figure 9.10: Graphic object panel

The graphic objects added in this menu belongs to the project. Each graphical modification applied to it involved the objects too. For example, if the project is rototranslate, the objects are rototranslate too. Otherwise, if objects are added in the node as explained in Graphic Objects, they are unlinked from the project, so they do not move.

CHAPTER

TEN

ELECTRODES

This element is used to display and manage the electrodes used for the data acquisition.

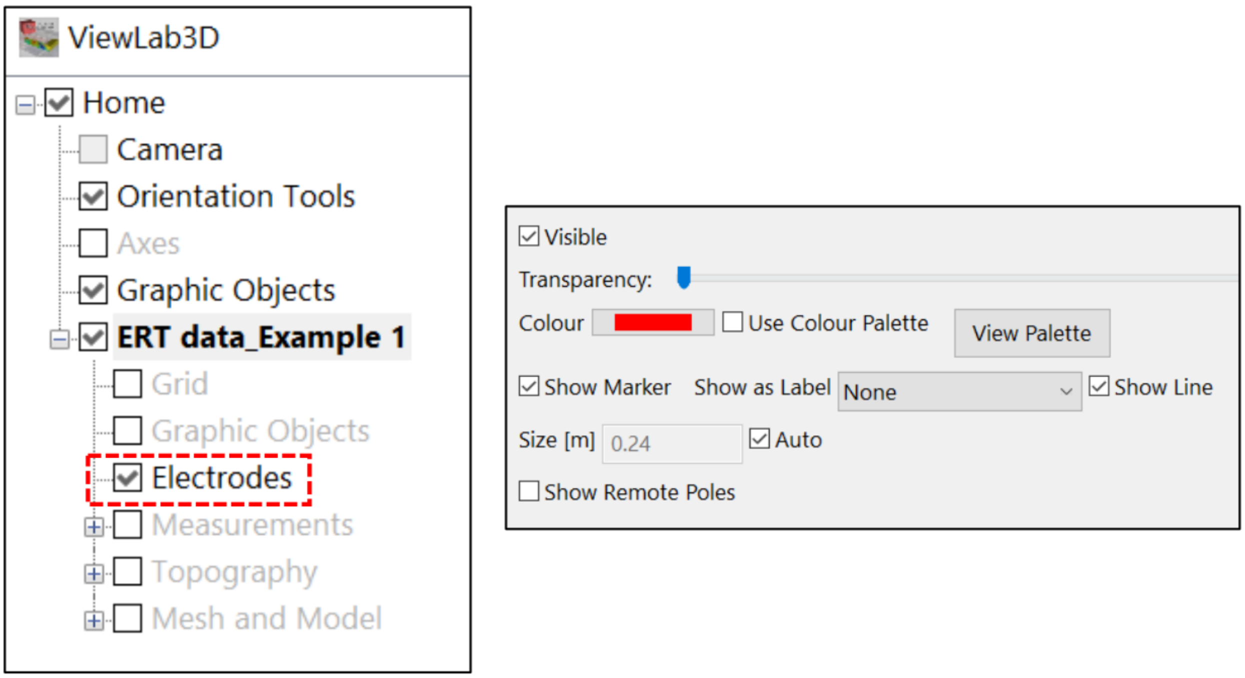


Figure 10.1: Electrodes Panel

As for other previously exposed menu items their *visibility* and *Transparency* can be selected. In addition, there are other dedicated tools, which are described below.

10.1 Colour

It is possible to choose the colour of the electrodes, by default it is red. When clicking on the *Colour* button, the colour panel will open, and is possible to choose another colour (Figure 10.2).

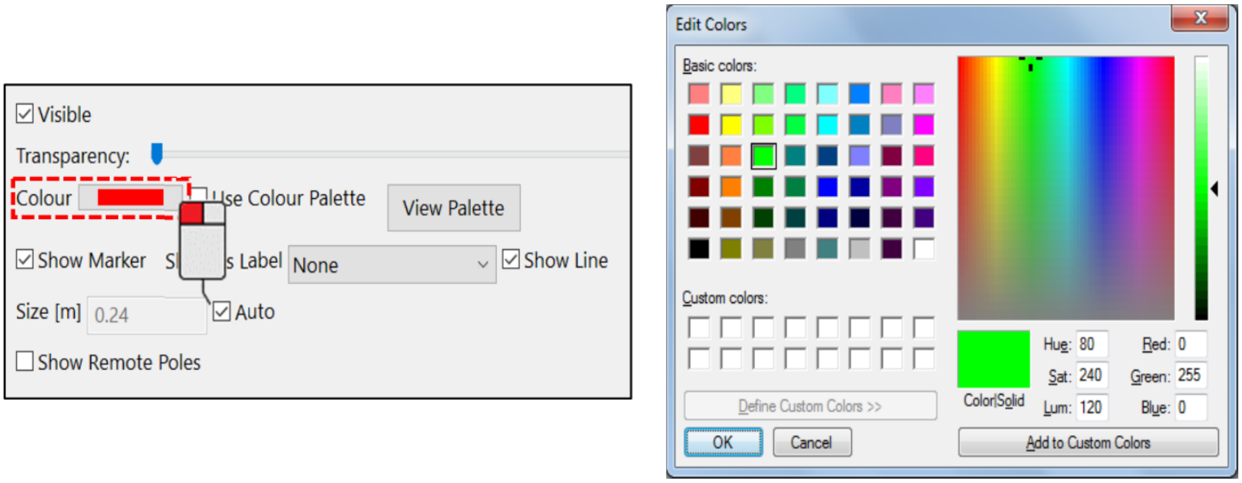


Figure 10.2: Electrode colour setting

Alternatively, in those cases where additional cables are loaded in the same project, it is possible to associate one specific colour to each cable, to easily recognize them. To do this, click on the *View Palette* button (Figure 10.3).

Figure 10.3: Electrodes palette setting

Each colour of the palette belongs to one cable (Figure 10.3). In Figure 10.3 shows the default palette. It is possible to customize it by changing the number of bars and their colours. In the following example, the third bar is deleted and a yellow one is added as the last colour of the sequence. With the *Delete Selected* button, the clicked bar is deleted (3 in Figure 10.4) and with the *Grow* button a bar is added to the end of the sequence. By clicking on a bar the colour can be chosen (yellow in Figure 10.4).

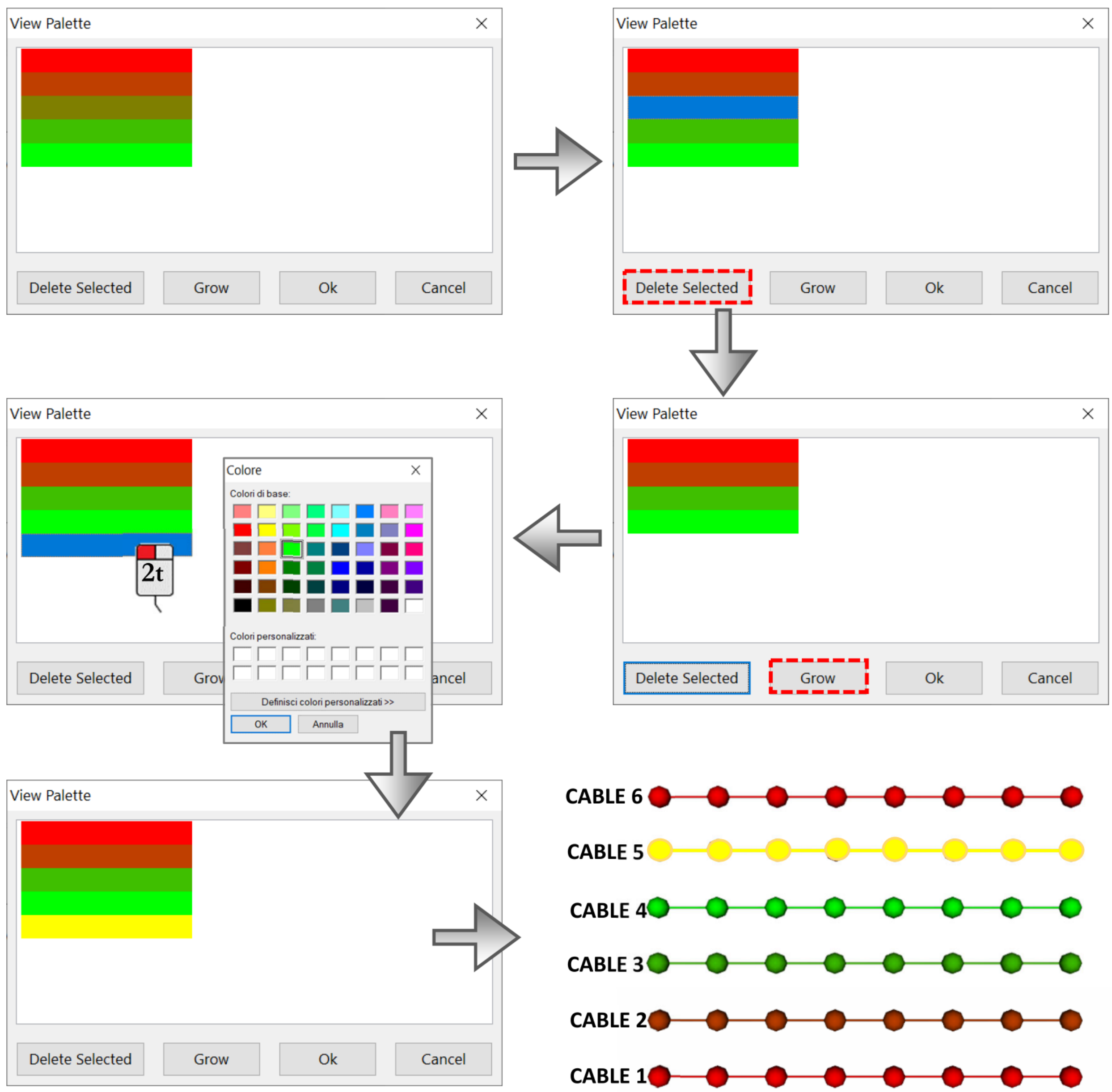


Figure 10.4: Changing colour to electrodes palette

In the following example shows the colour selections changed (Figure 10.5).

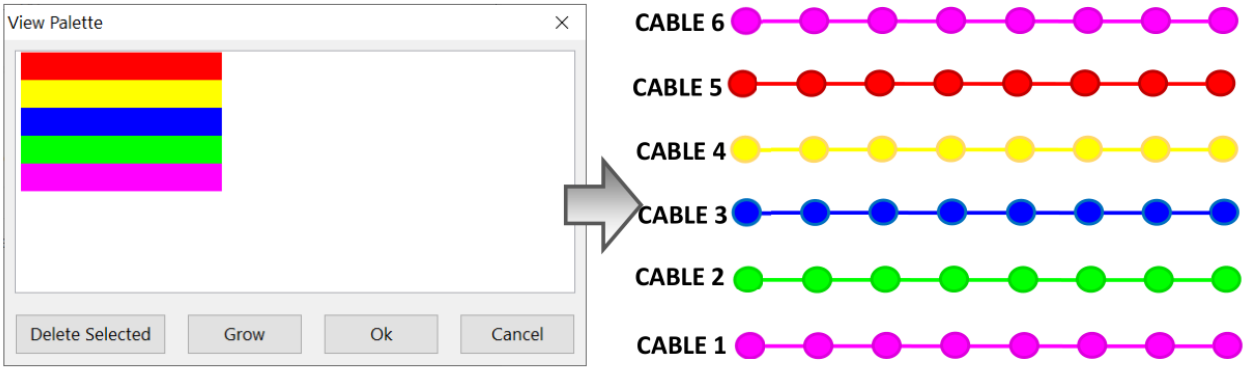


Figure 10.5: New palette colours

If the number of cables are bigger that the number of palette bars, the cycle of colours starts again (in fact in the examples the first and the last cable are of the same colour, pink).

10.2 Show Marker

If this checkbox is selected, the electrodes positioned in the scene are visible, otherwise they are present in the dataset but they are not visible in the 3D scene (Figure 10.6).

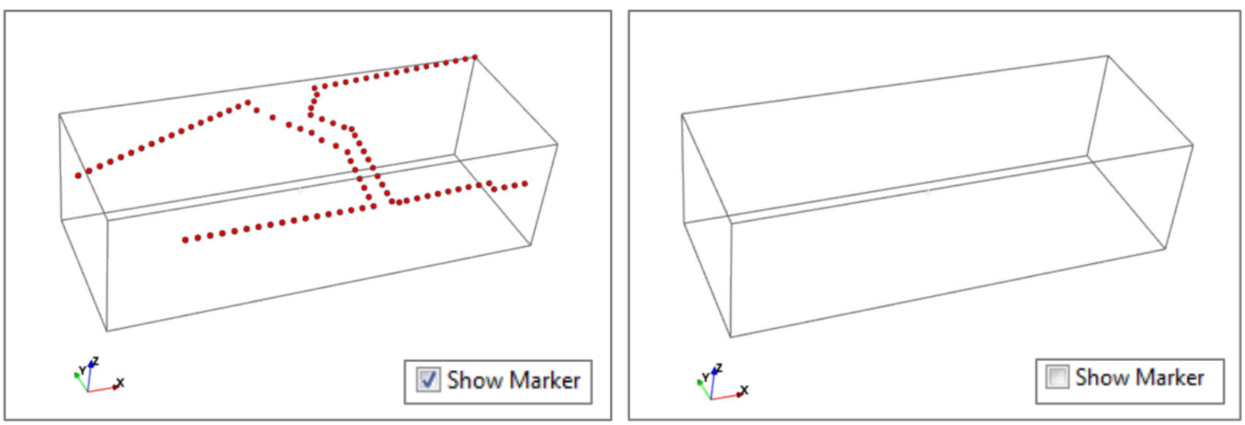


Figure 10.6: Sub-node electrodes; in A the electrodes are visible, in B not

10.3 Show Line

If this checkbox is selected, a line that connects the electrodes is visible, otherwise it is not, as it is shown in Figure 10.7.

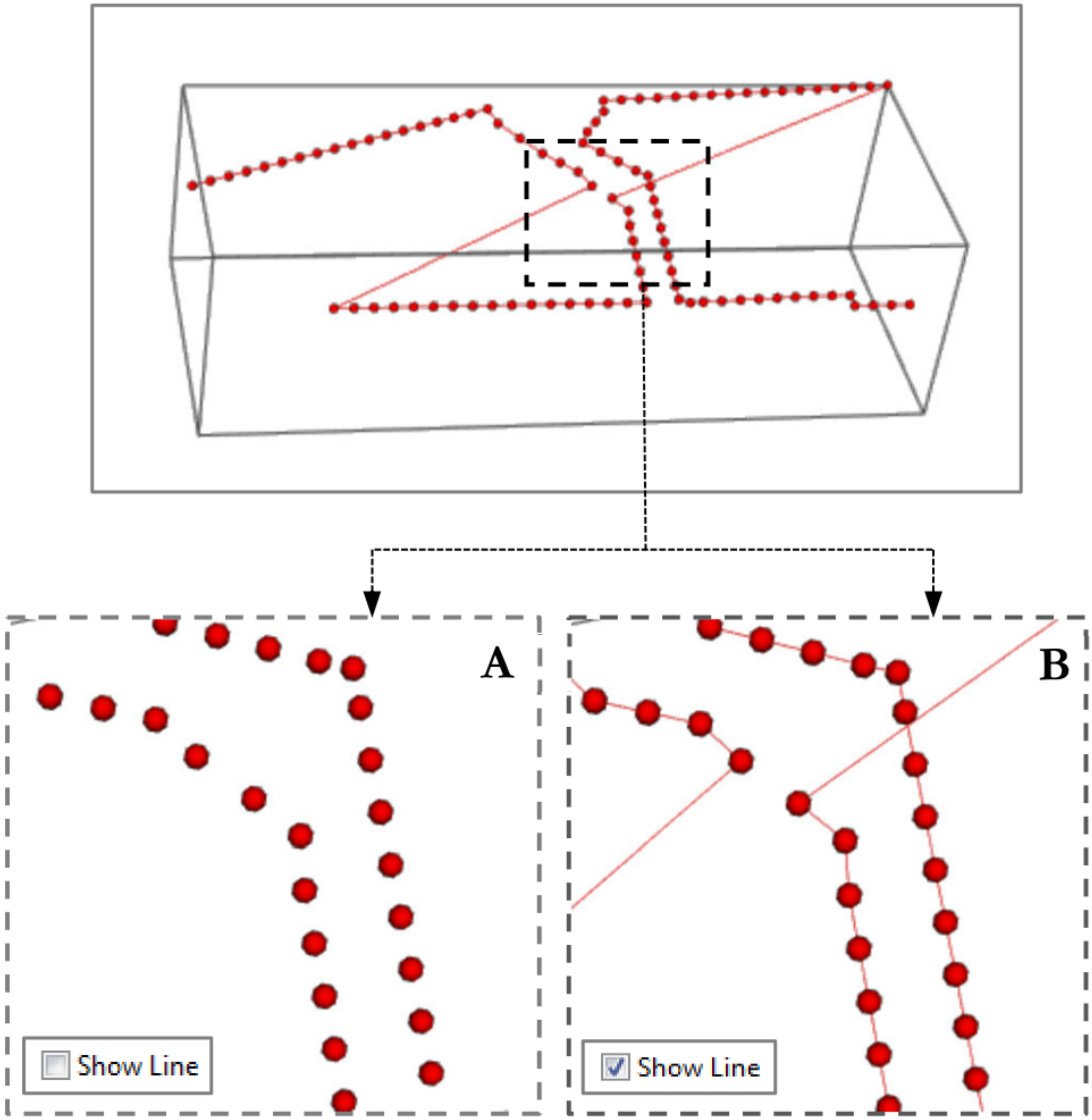


Figure 10.7: Detail of electrodes with (at right, B) and without (at left, A) the line between electrodes

10.4 Show Label

If this checkbox is selected, the labels of the electrodes are visible, otherwise they are not. Click in the drop-down box (Figure 10.8) to choose between 3 types of labels:

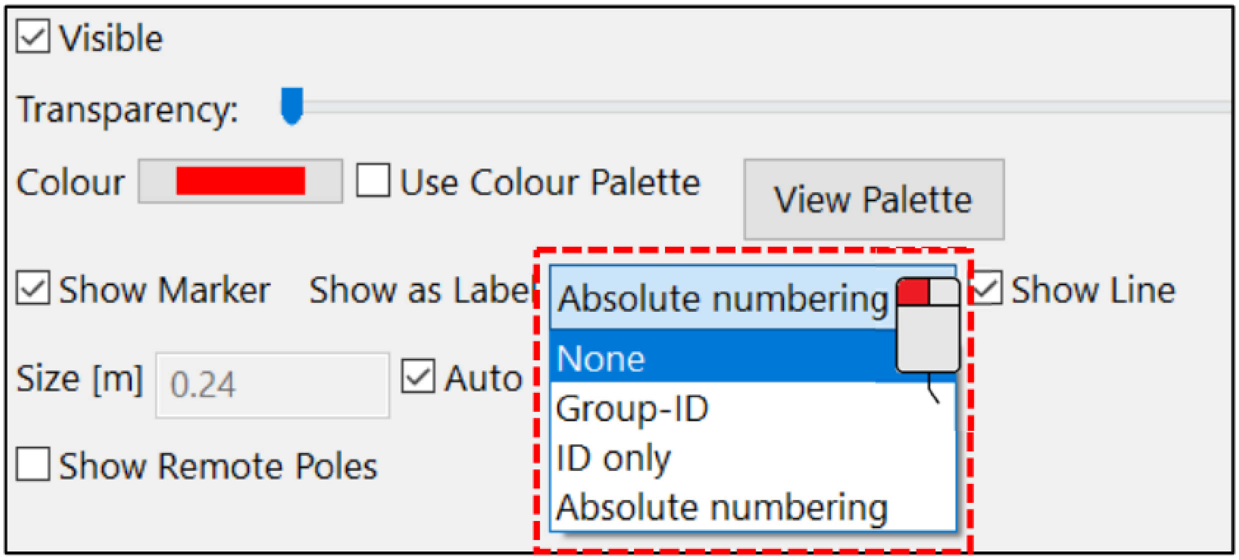


Figure 10.8: Labels menu

- *Group-ID*: this label contains the name associated with each electrode during the creation of the project, and it is composed of the "Group" (PD in the following example) and the identification number "ID" (from 1 to 96 in the following example) of each electrode (Figure 10.9);
- *ID only*: this label contains just the identification number associated with each electrode during the creation of the project (Figure 10.9);
- *Absolute numbering*: this label contains the numeration of electrodes; it may coincide with the ID, as in the example in (Figure 10.9).

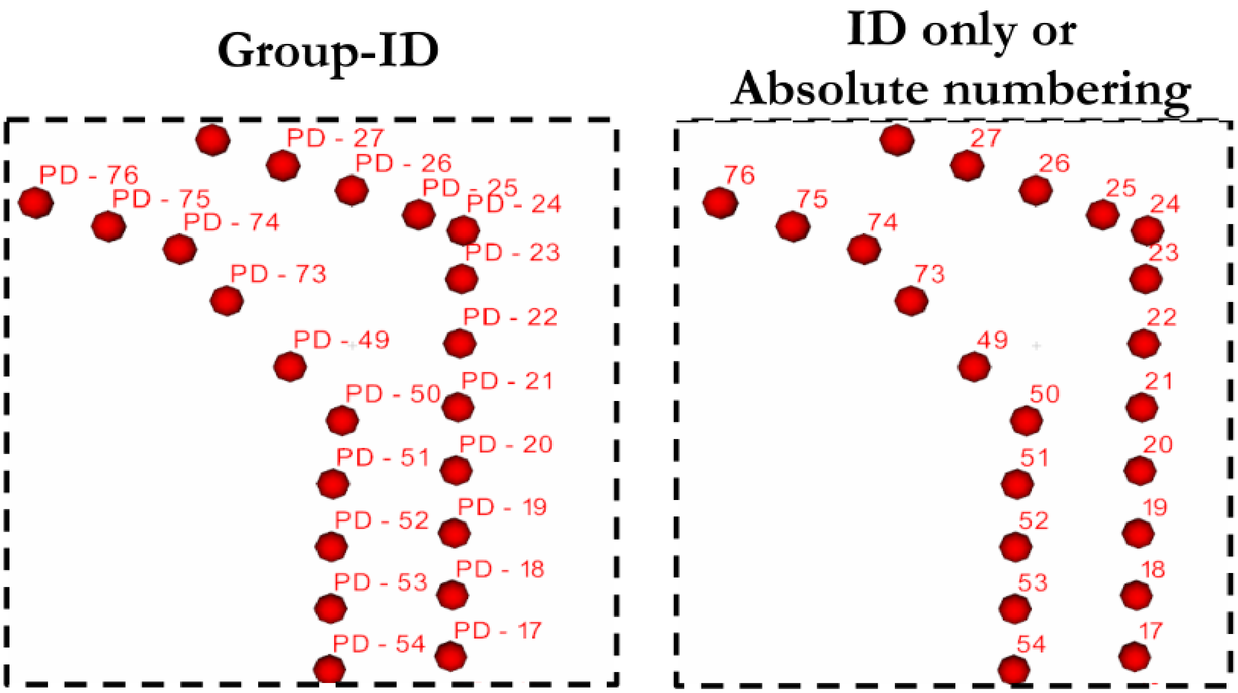


Figure 10.9: Example of different kind of electrodes labels

10.5 Size

By default, the option "*Auto*" is checked, meaning that the dimension of the electrodes is automatically determined. It is possible to change it by unchecking the *Auto* checkbox and using the dedicated tools. Use the slider or manually insert the size number (in green in Figure 10.10, 0.5 by default) between a minimum and maximum value in meters (0 and 30 in this case, in the red circles in Figure 10.10). Within this interval, it is possible to set a relative minimum and maximum value (in orange in Figure 10.10), by writing the corresponding numbers in the dedicated boxes. This way the extreme values of the slider change from 0-30 to the new selected values.

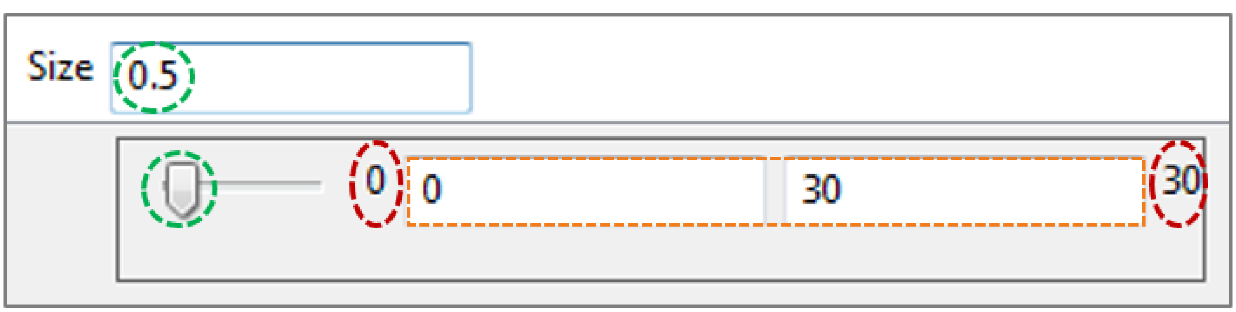


Figure 10.10: Tool for electrodes size setting

For example, if the minimum size should be 5 and the maximum 15, writing those numbers in the first two dedicated boxes (in orange in Figure 10.11) the slider will have a value of size 5 when it is totally on the left, 15 when it is totally on the right, and an intermediate value along the sliding bar (Figure 10.12).

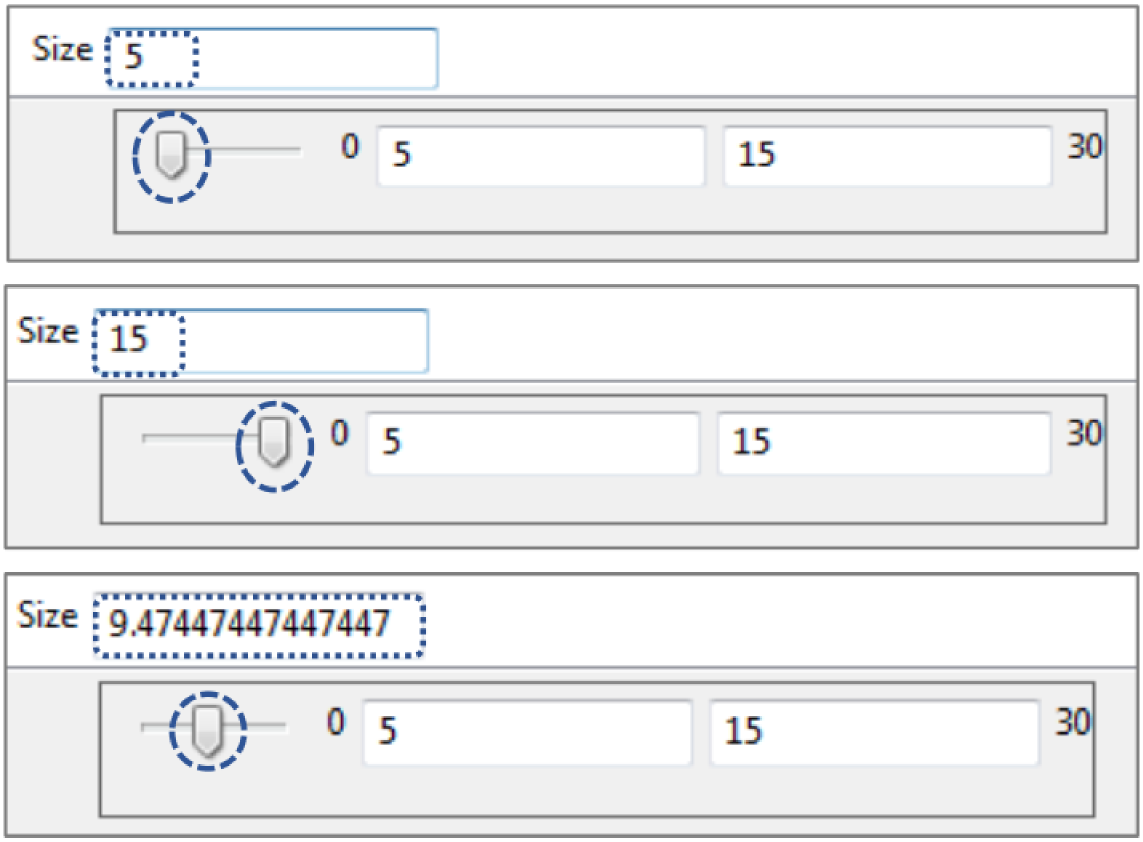


Figure 10.11: Example of variation of extreme values of the sliding bar, from 0-30 to 5-15

As an example the electrodes are represented in two different sizes in Figure 10.12.

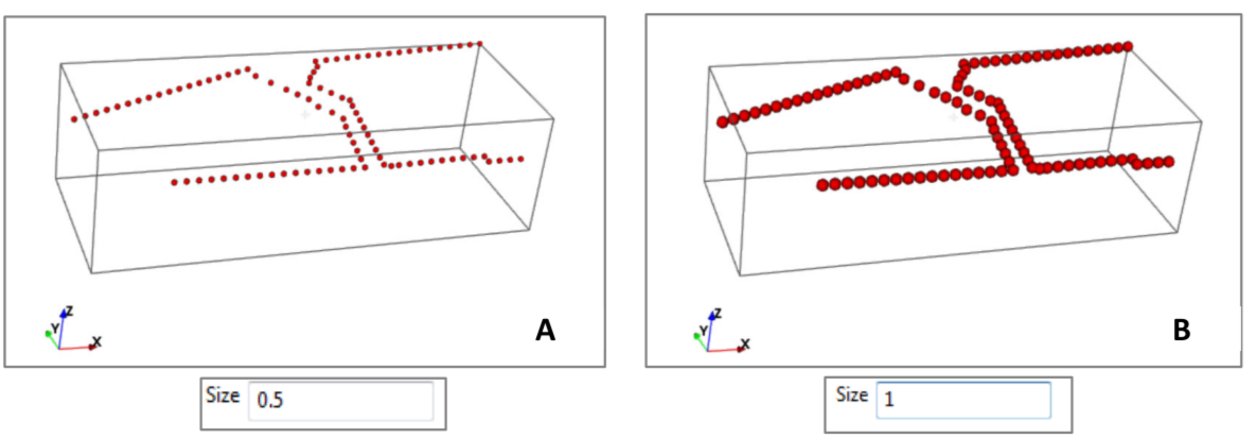


Figure 10.12: Sub-node electrodes, example of two electrodes sizes, 0.5 in A and 1 in B

10.6 Show remote pole

If the data acquisition uses a remote pole it can be visualized through this checkbox. For a better representation of the scene use the auto centre button. It is advisable to not visualize the remote pole, because it is far from the other electrodes and including it into the scene the investigated area may appear too small.

CHAPTER

ELEVEN

MEASUREMENTS

This element is used to display and manage the measurements available in the dataset, acquired and calculated data.

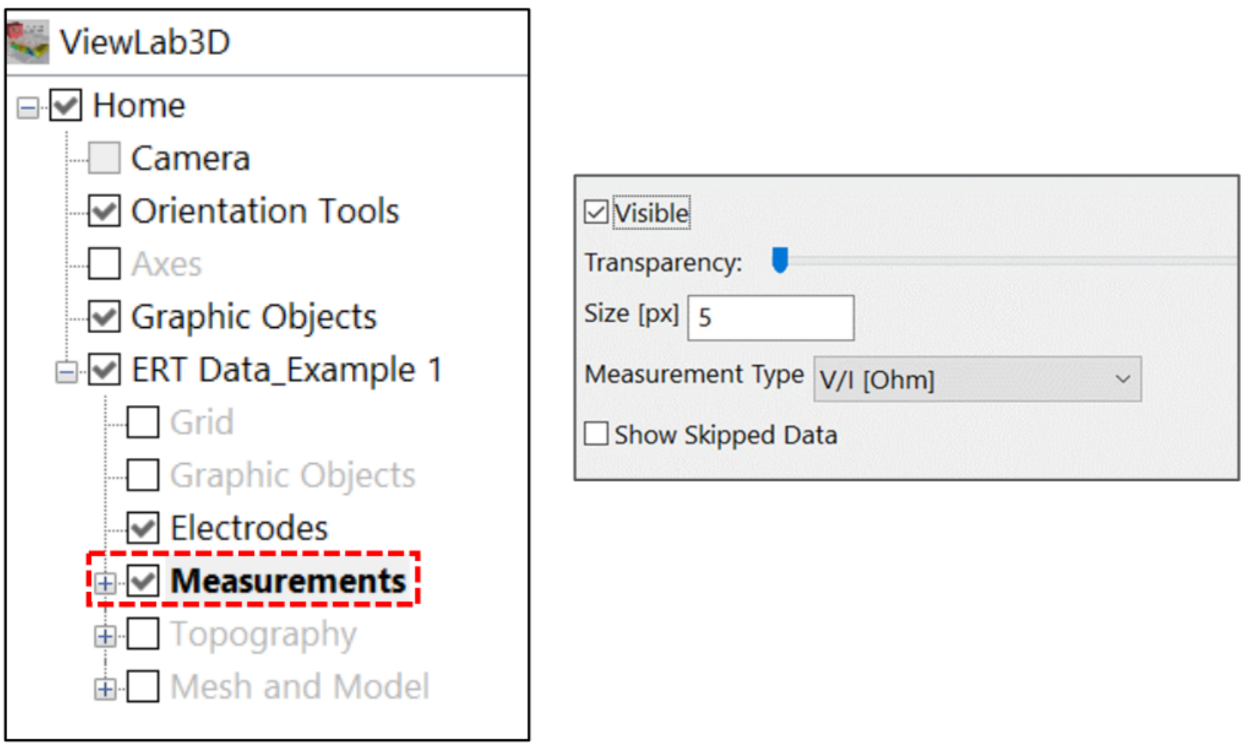


Figure 11.1: Measurement panel

As for other options exposed in the previous paragraphs, even for measurements the user can select their *visibility* and set the level of *Transparency*. In addition, there are others specific new options, which are described below.

11.1 Size

It is the size of the points which represent every single measurement in the 3D space. It is expressed in pixels [px].

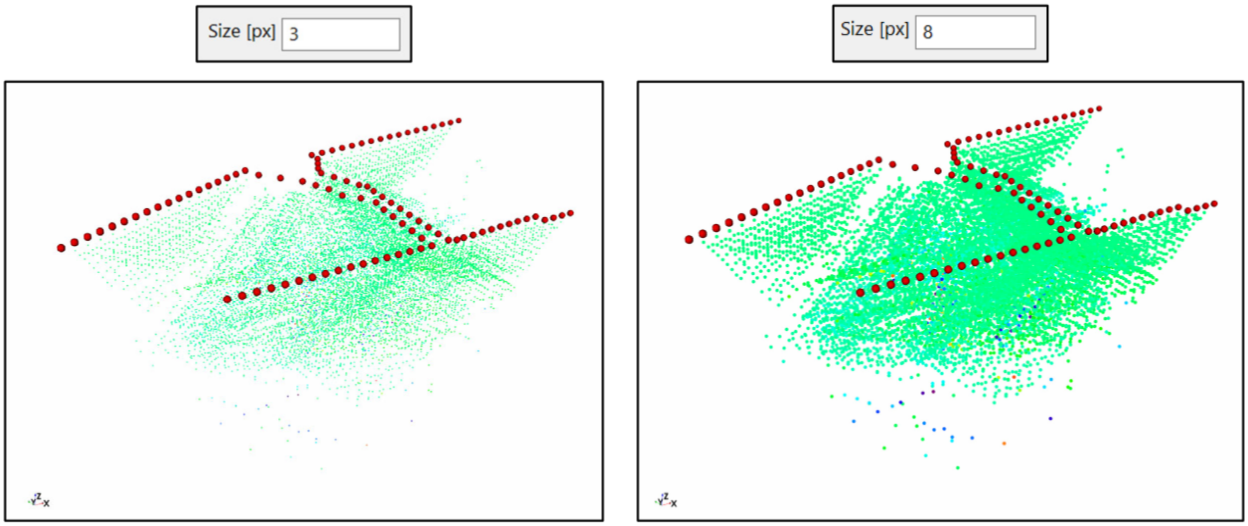


Figure 11.2: Different size of Apparent Resistivity points

11.2 Measurement Type

Choose the type of measurements to visualize using this pop-up menu, available measurements are:

- V/I [Ohm]
- Apparent Resistivity [Ohm*m]
- IP [mV/V]
- K [m]
- V[mV]
- I [mA]
- Standard Deviation V/I [Ohm]
- Standard Deviation IP [mV/V]
- Calculated V/I [Ohm]
- Calculated Apparent Resistivity [Ohm*m]
- Calculated IP [mV/V]
- Calculated Standard Deviation V/I [Ohm]

- Calculated Standard Deviation IP [mV/V]

11.3 Colour Scale

A node Colour Scale is also available to customize how the value needs to be rendered as colour.

CHAPTER

TWELVE

TOPOGRAPHY

This option loads and manages the topography. Adding topographical information can be useful for a better representation and analysis.

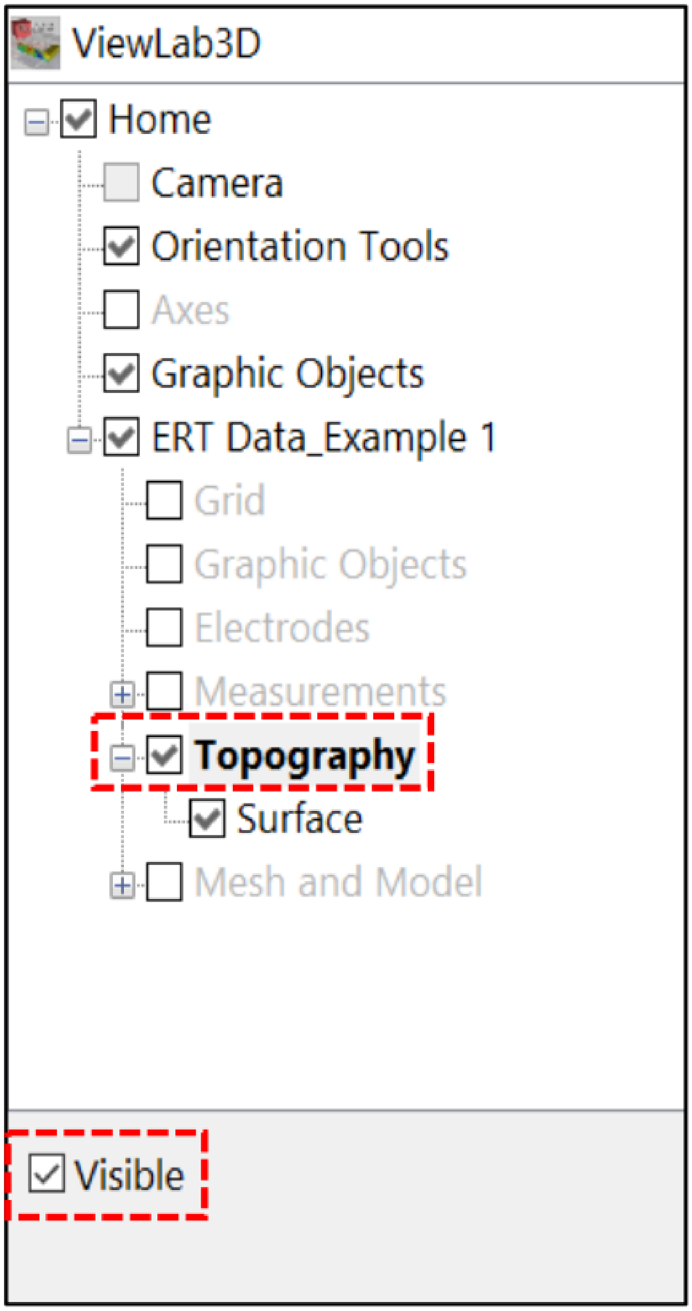


Figure 12.1: Topography properties

To make the topography visible check *Visible* on the Topography Panel.

The *Topography* node has also a second child node named *Surface*. The *Surface* option is used to manage the way the topography is displayed, as points or as a surface that interpolates each point. The Surface panel is composed of two blocks:

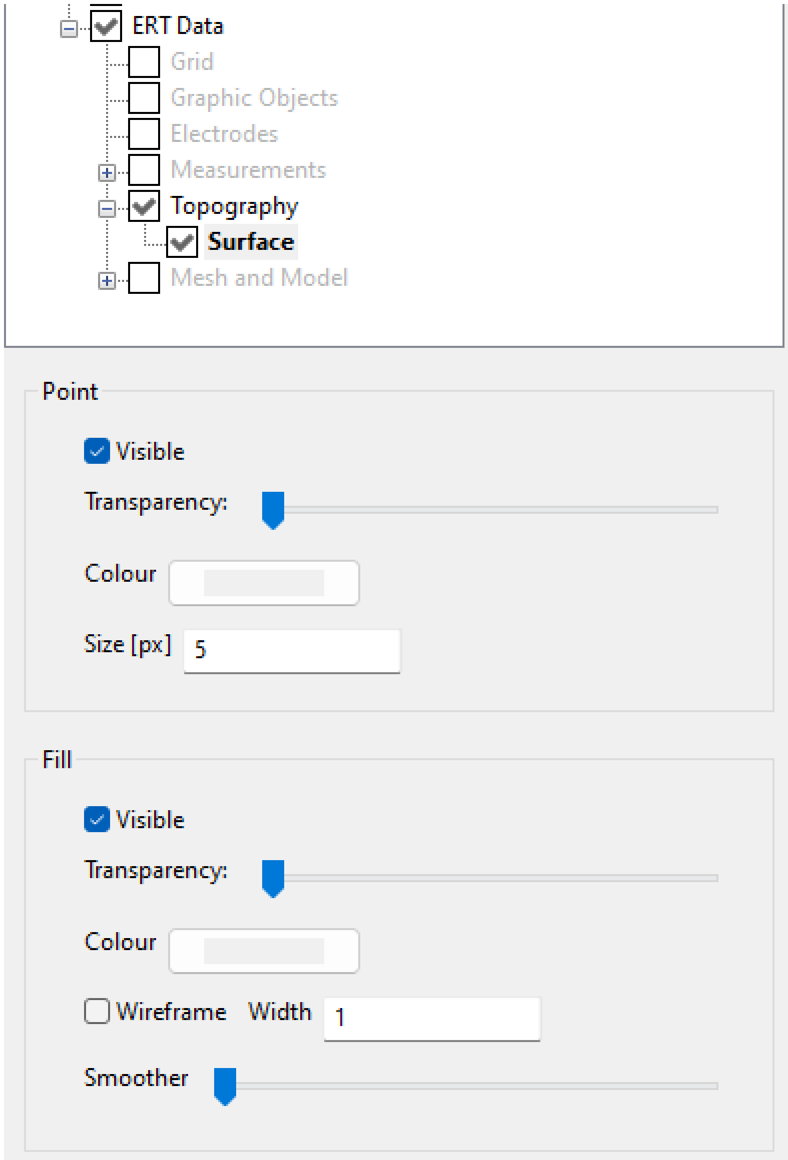


Figure 12.2: Surface properties

12.1 Point

It manages the visualization mode of the points in the topography file. Each coordinate triplet corresponds to one point in the *Surface* visualization.

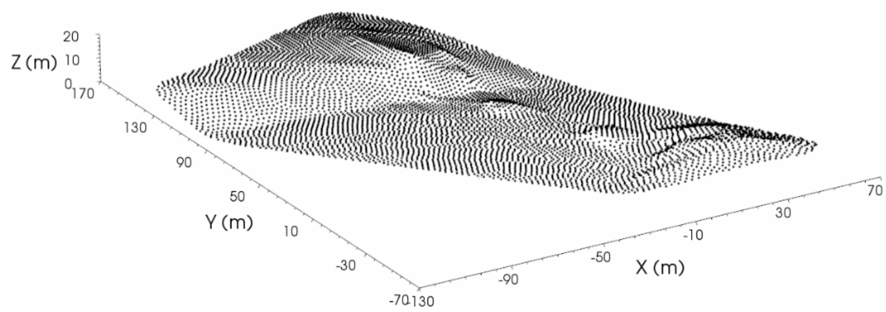


Figure 12.3: Example of Surface visualized as points

Checking *Visible* in the Point Window visualizes the topography in the 3D scene as simple points.

It is possible to alter the level of *Transparency* through the slider, the *Colour* and the *Size* of the topographic points (Figure 12.4).

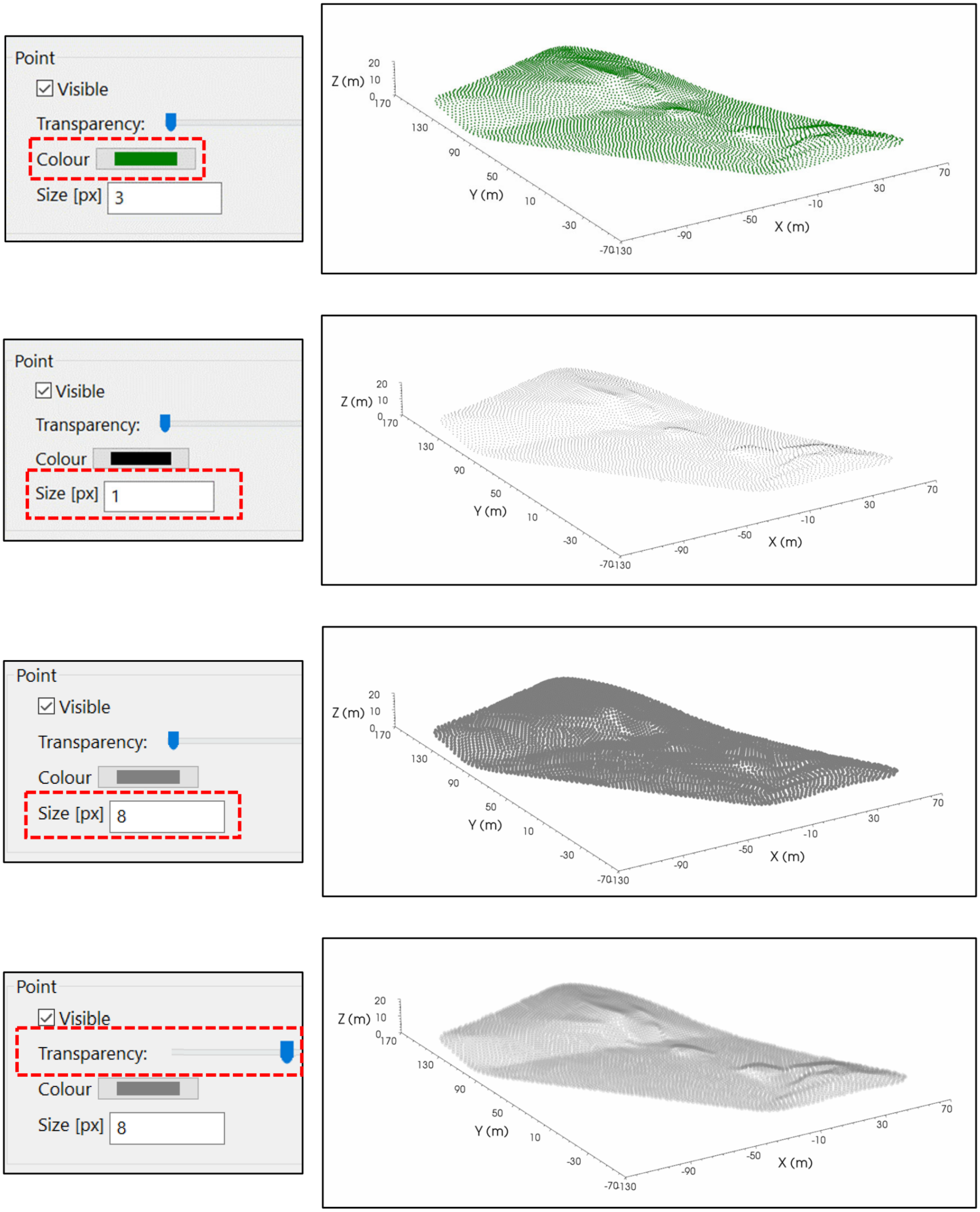


Figure 12.4: Examples of different sizes of surface points

12.2 Fill

manages the representation of the topographic points as a surface that interpolates each point. In addition to the same *Point* window options, it has some additional elements, highlighted in red in Figure 12.5.

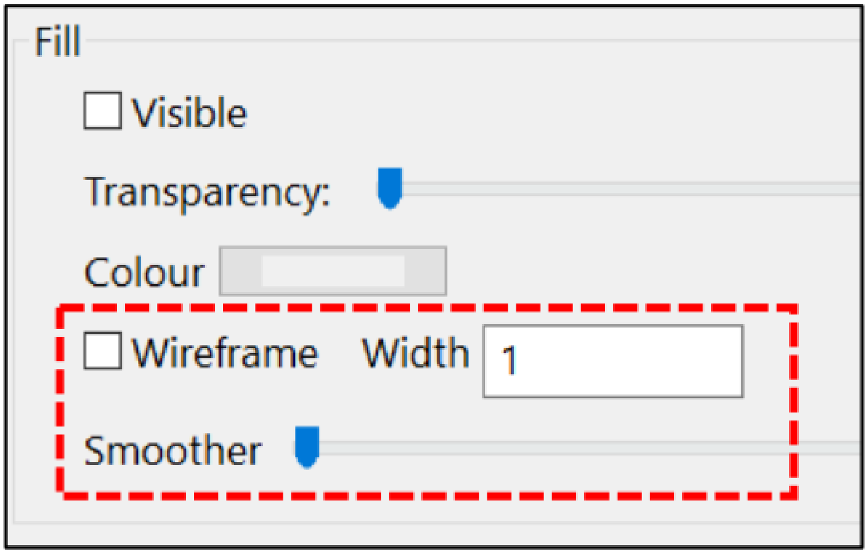


Figure 12.5: Fill surface panel

If the checkbox “*Wireframe*” is checked, the topography is visualized as a 3D model in which only lines and vertices are represented.

Otherwise the entire surface is visualized (Figure 12.6).

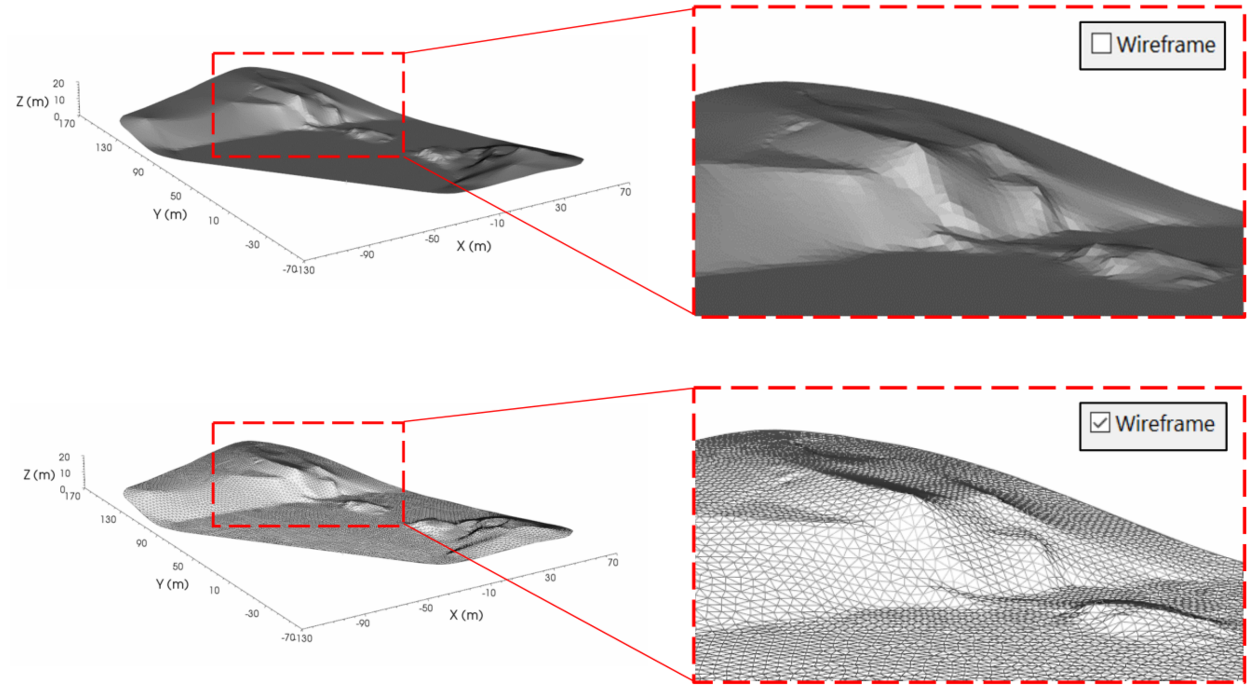


Figure 12.6: Surface visualized as wireframe

If the *Wireframe* mode is active, it is possible to choose the *Width* of the lines that constitutes the topography, typing the desired number in the proper box.

In Figure 12.7 the default value 1 is changed in 3.

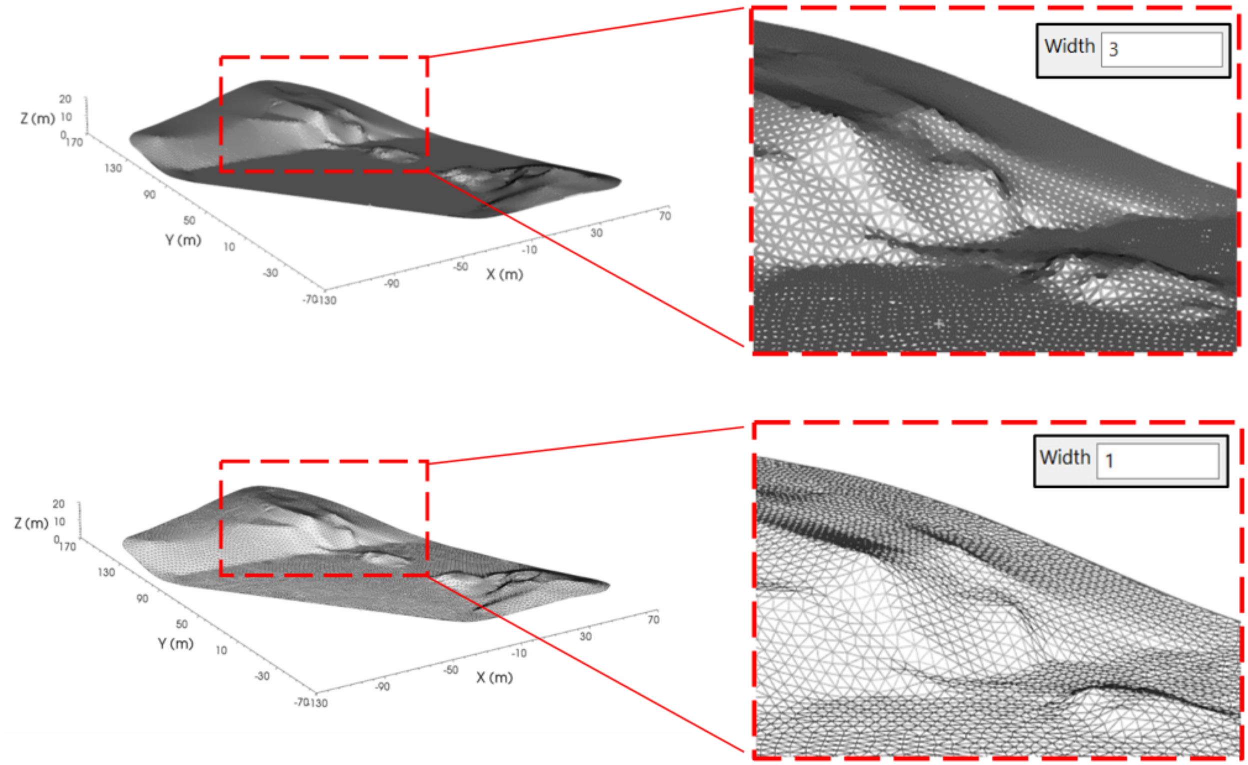


Figure 12.7: Examples of different size of wireframe width

The *Smoother* slider allows to smooth the topography. It is advisable to not make it to smooth in order to not cause a slowdowns of the software.

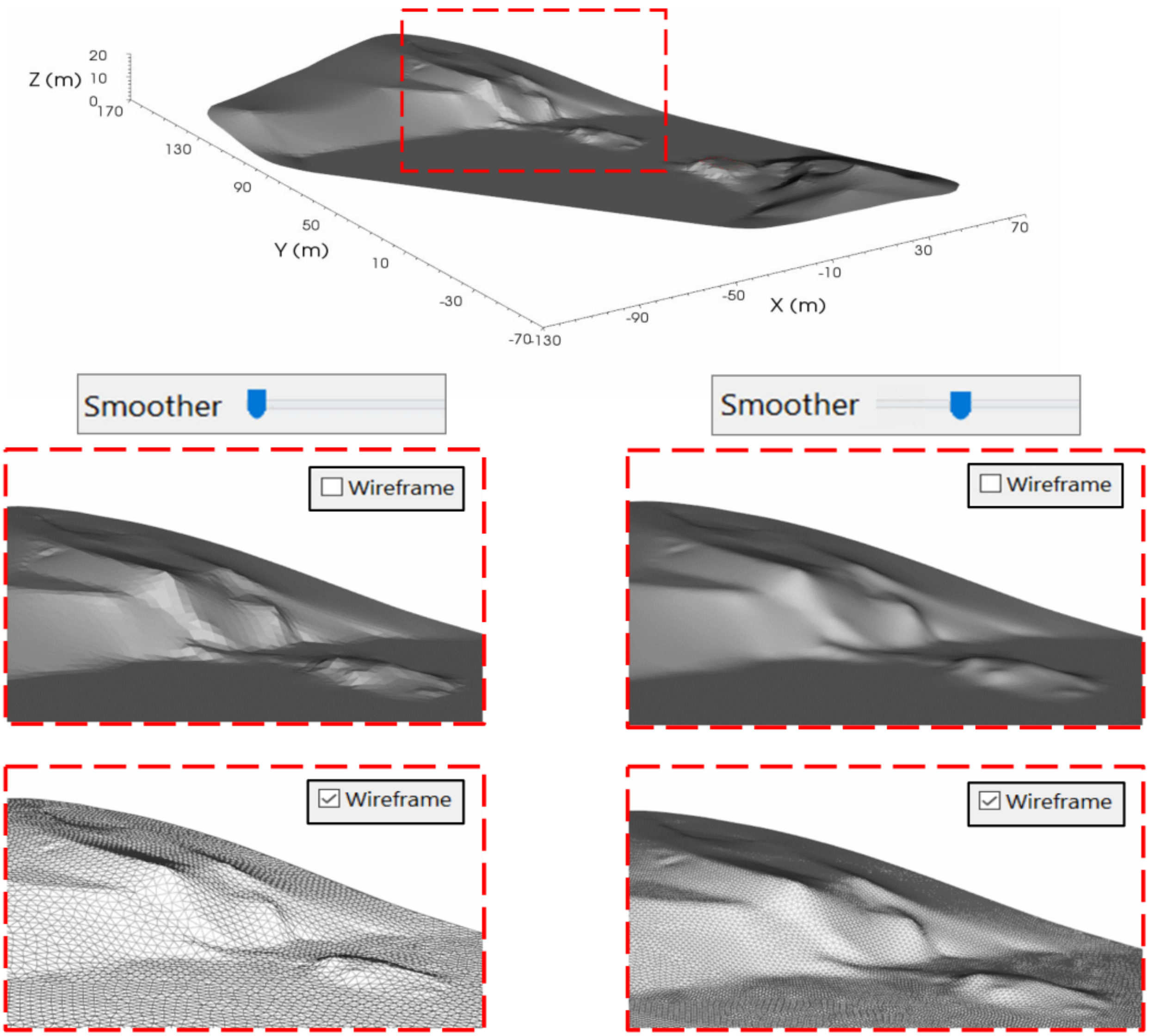


Figure 12.8: Example of surface topography smooth, with and without wireframe mode active

CHAPTER THIRTEEN

MESH AND MODEL

This menu option has the purpose of modelling the subsurface with a 3D grid (mesh) and to assign physical properties values(model) to the points of this half-space.

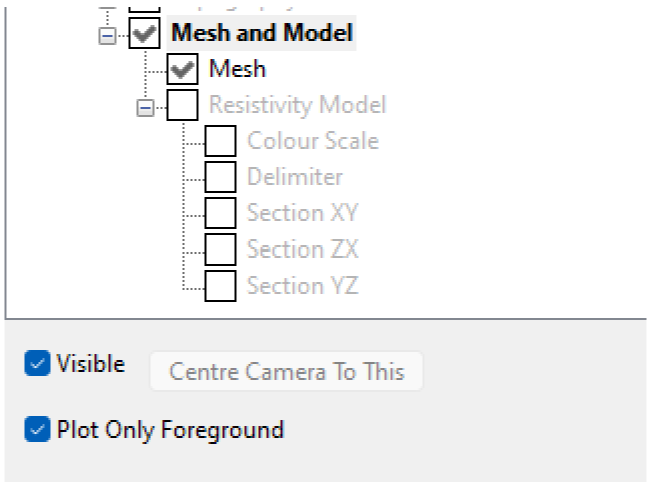


Figure 13.1: Mesh and Model properties

If "*Plot Only Foreground*" is checked the Background region is not visible; otherwise the entire mesh is visualized.

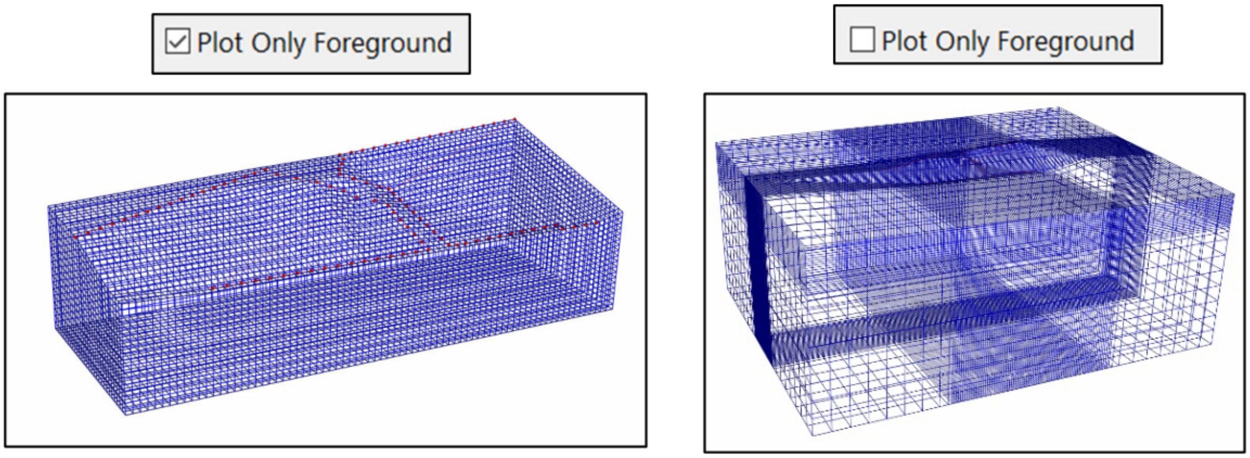


Figure 13.2: On the left, mesh with “Plot Only Foreground” flagged; on the right, mesh with “Plot Only Foreground” not flagged, so also the background region is displayed.

There can be some child nodes, which are explained below.

13.1 Mesh

This option manages the way of the mesh representation. The window has the following items:

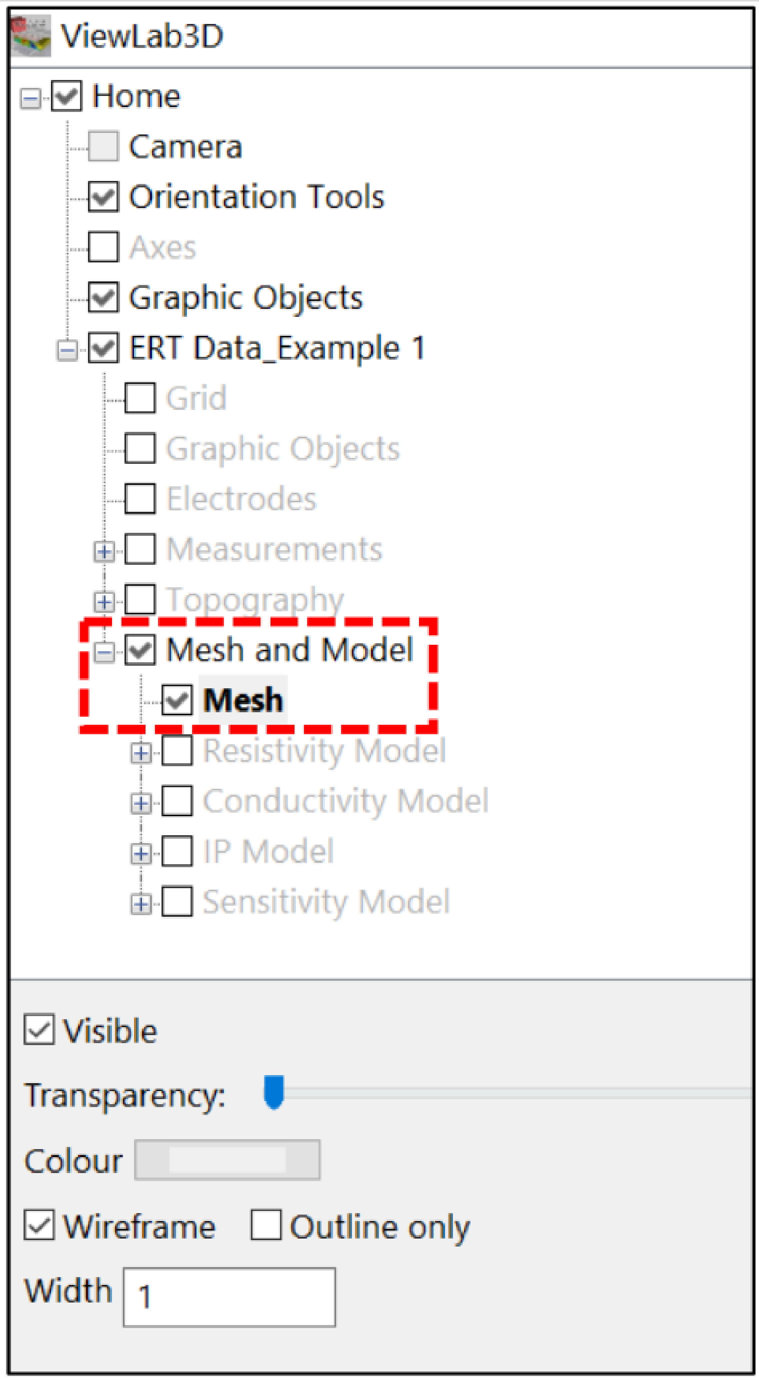


Figure 13.3: Mesh panel

13.1.1 Visible

The checkbox controls the visibility of the mesh.

13.1.2 Transparency

Through the slider the transparency of the mesh, as it is shown can be controlled (Figure 13.4).

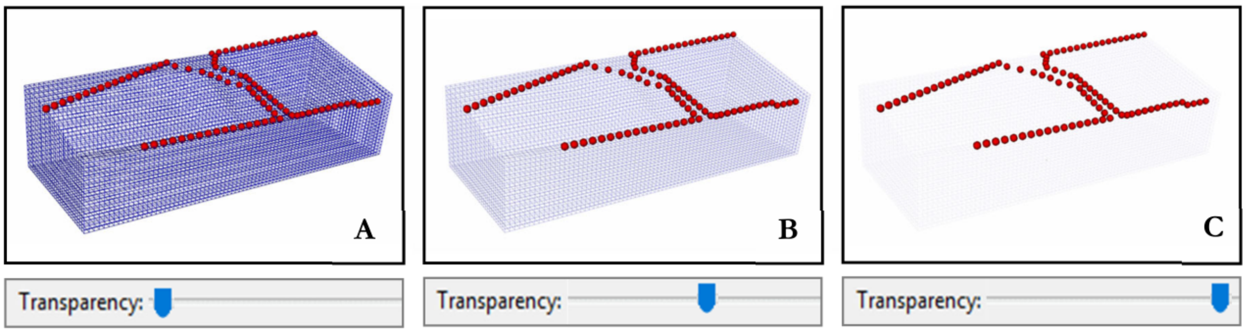


Figure 13.4: Example of different levels of transparency of the mesh

13.1.3 Colour

By default the mesh is grey, but it can be changed to any colour. Figure 13.5 shows a change from grey to green.

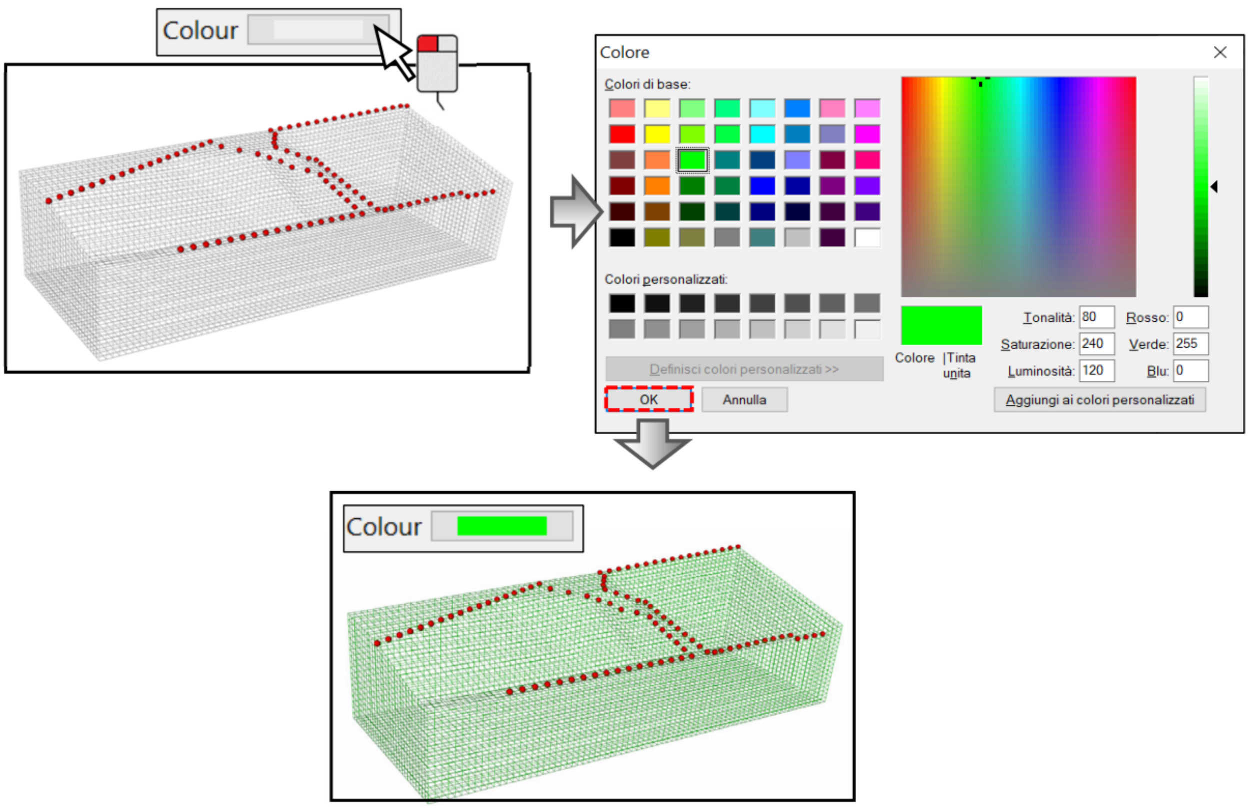


Figure 13.5: Example of mesh colour change

13.1.4 Wireframe

If the *Wireframe* box is checked the mesh is visible as lines. If *Outline only* is checked too, the mesh is visualized with just the external lines (case A in Figure 13.6). If *Outline only* is not checked the mesh is visualized with every cell that constitutes the entire volume (case B in Figure 13.6). If both the boxes are unchecked the volume is visualized as a "solid" body (case C in Figure 13.6).

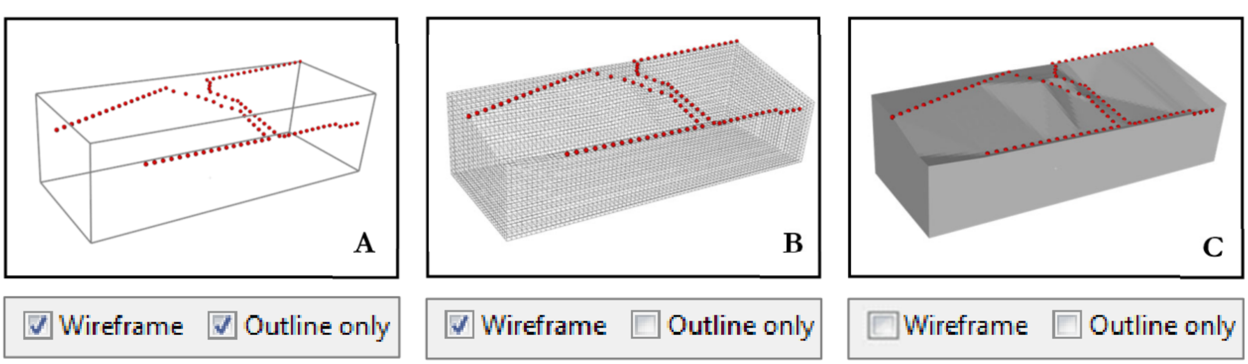


Figure 13.6: Example of different combination of 'wireframe' and 'outline only' mesh visualization mode

13.1.5 Width

If *Wirefram* e is selected, the width of the mesh can be chosen by typing a number from 1 to 10 or through the slider which appears clicking on the box. Two different outline widths are shown in Figure 13.7.

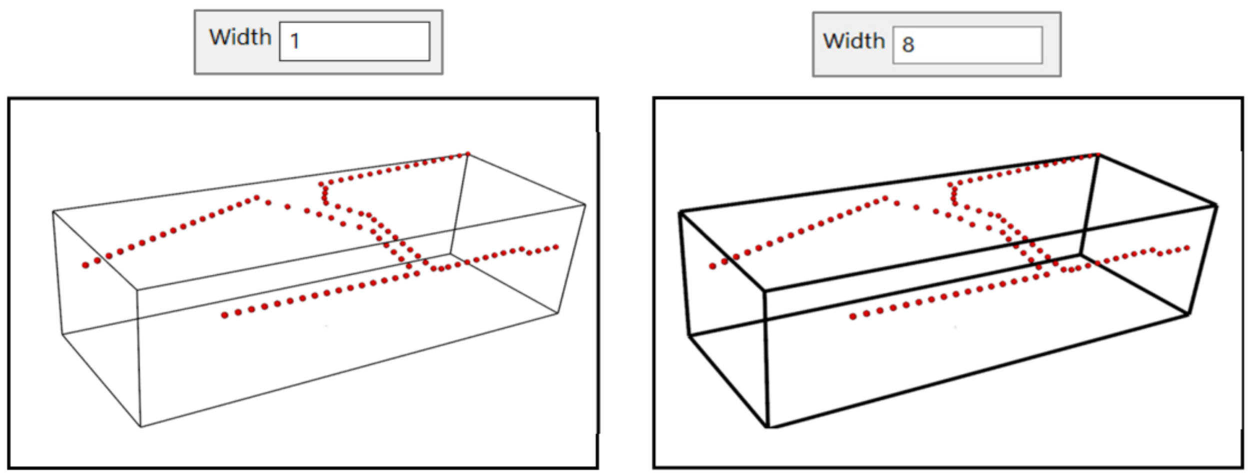
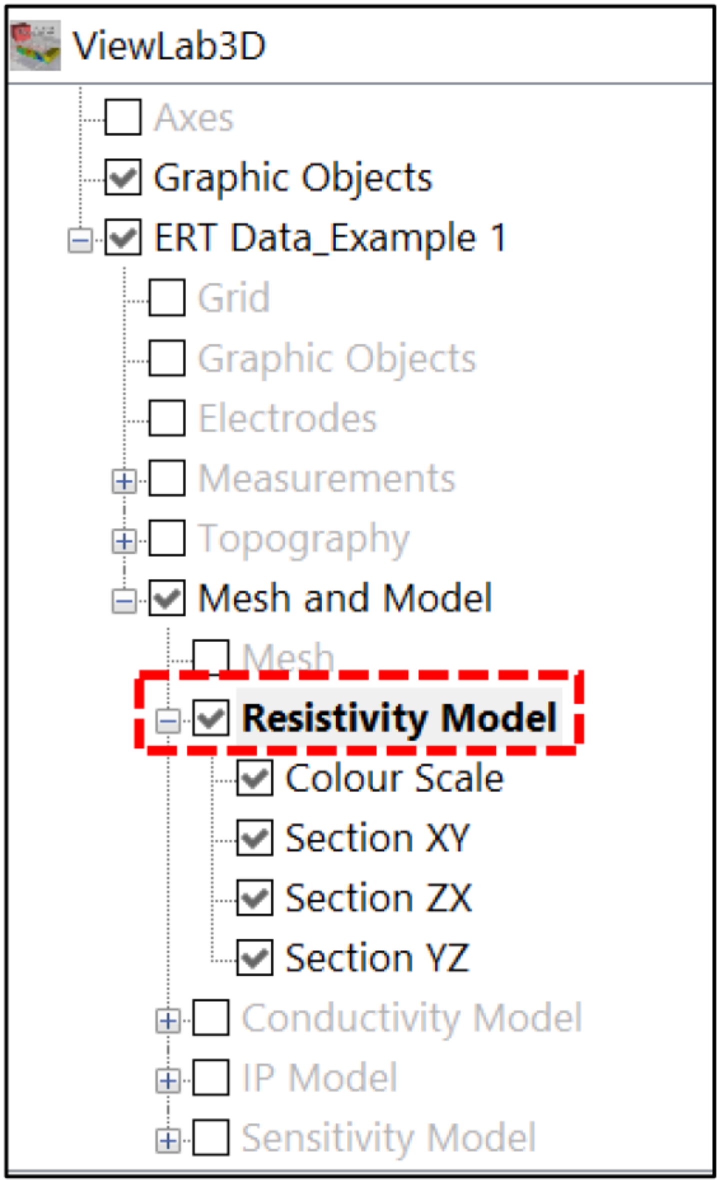


Figure 13.7: Example of different mesh widths

13.2 Resistivity Model

The *Resistivity Model* quantifies how strongly the investigated body contrasts the flow of electric current. The Resistivity Model is present in the project after the inversion process has been run (Figure 13.8).



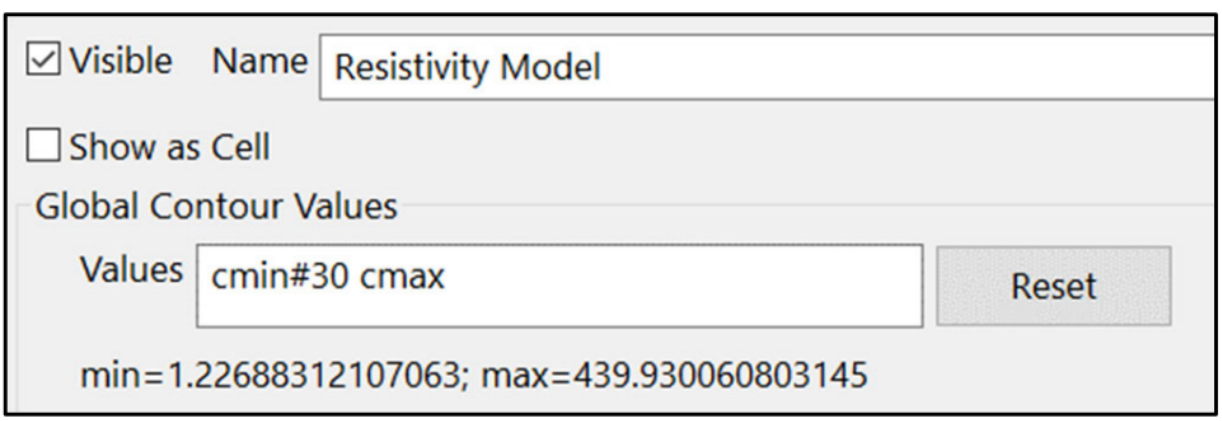


Figure 13.8: Resistivity Model panel

13.2.1 Visible

This checkbox controls the *Resistivity Model* visibility;

13.2.2 Name

By default, the node name is labeled “Resistivity Model”, but it possible to change it by typing the desired new name in this box;

13.2.3 Show as cell

If this checkbox is selected, the model is shown composed by cells. Otherwise it is represented as an interpolation of points. The model obtained in this last case is smoother (Figure 13.9).

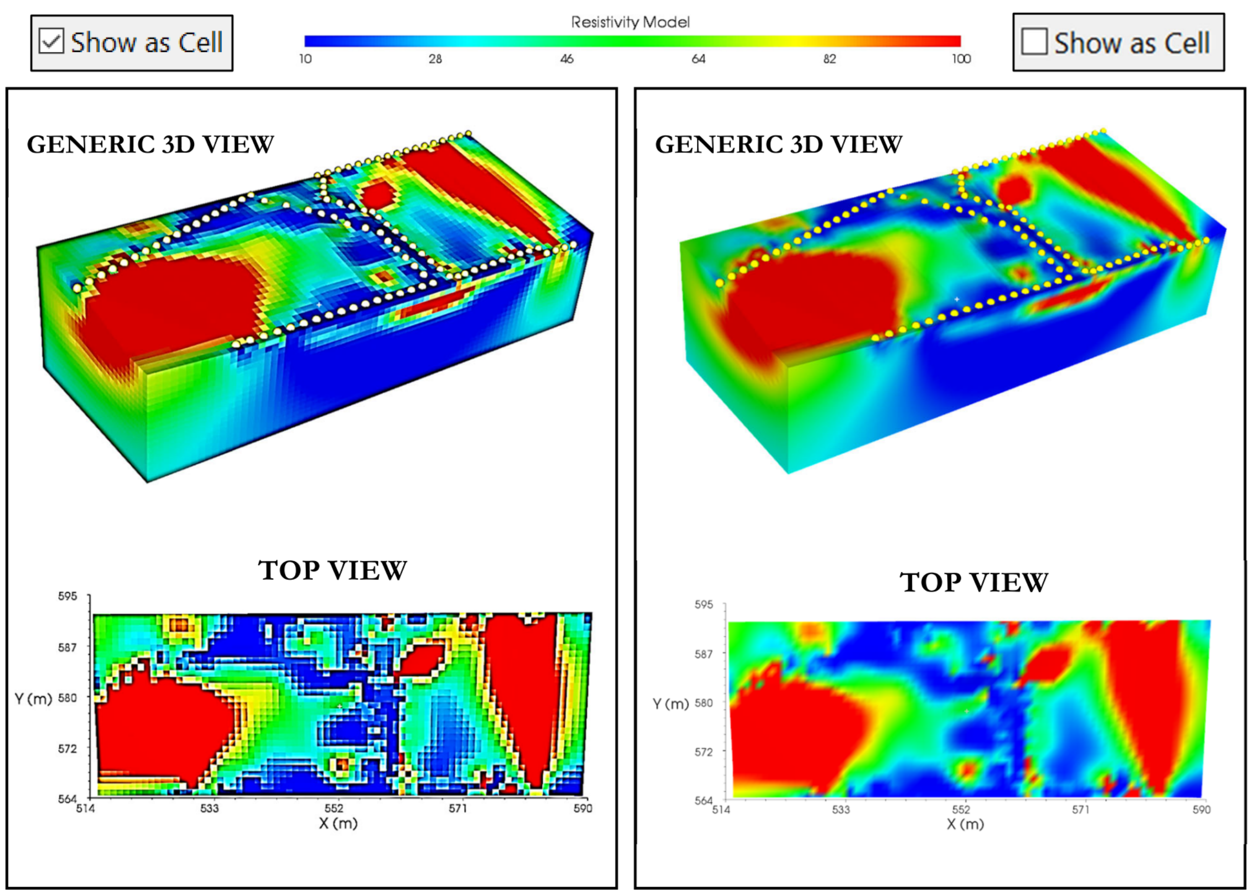


Figure 13.9: Examples of Resistivity Model with “show as cells” checked on the left and not checked on the right

13.2.4 Global Contour Values

Manages the contour lines of all sections inserted in the Resistivity Model. The maximum and minimum value to apply at the contouring and the step interval within one line and the other can be selected. In the lower part of the box the minimum and maximum value of the entire dataset is automatically reported.

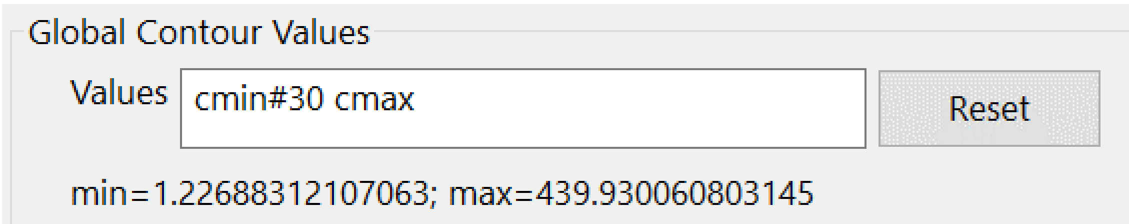


Figure 13.10: Resistivity Model sub-panel

These values are applied to each section, but it is possible to change them and set custom values to each section, as is shown in this section.

13.2.5 Sub nodes

- **Colour Scale**: allows to set the properties of the colour scale as described in the Colour Scale section.
- **Delimiter**: allows to show only a portion of the available mesh, see the *Delimiter* section.
- **Section XY-ZX-YZ**: there are three predefined sections, one for each main direction XY, XZ, YZ.

Clicking with the right mouse button on the *Resistivity Model* node it is possible to add some additional nodes. They are:

- **Sections**: see Model Section
- **Volume**: see Model Volume
- **Isosurface**: see Model Isosurface

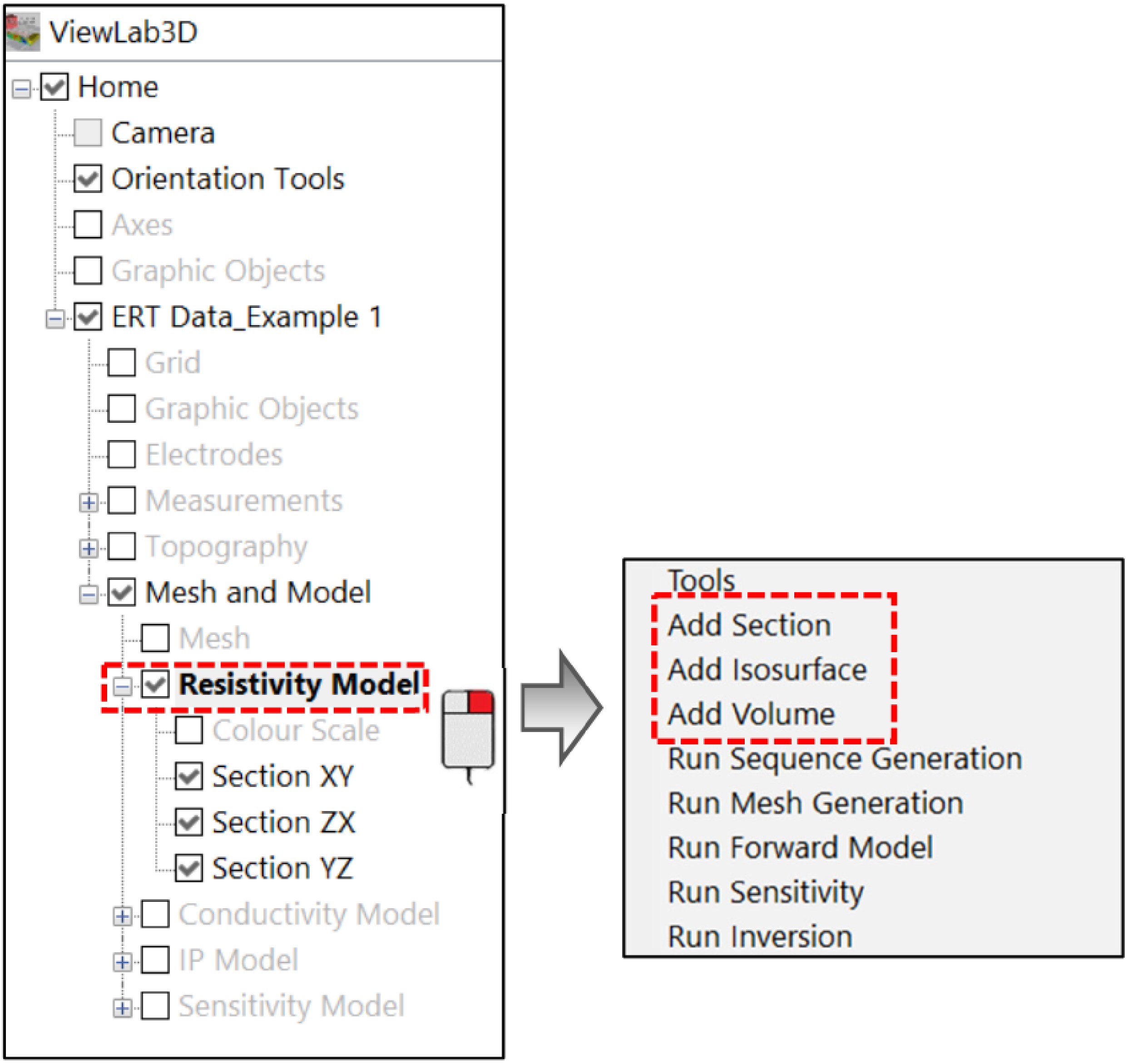


Figure 13.11: Add Section/Isosurface/Volume to the Resistivity Model

13.3 Examples

To close out this section we visualize more Resistivity Model examples in Figure 13.12 and Figure 13.13 to show some of the many data representation possibilities.

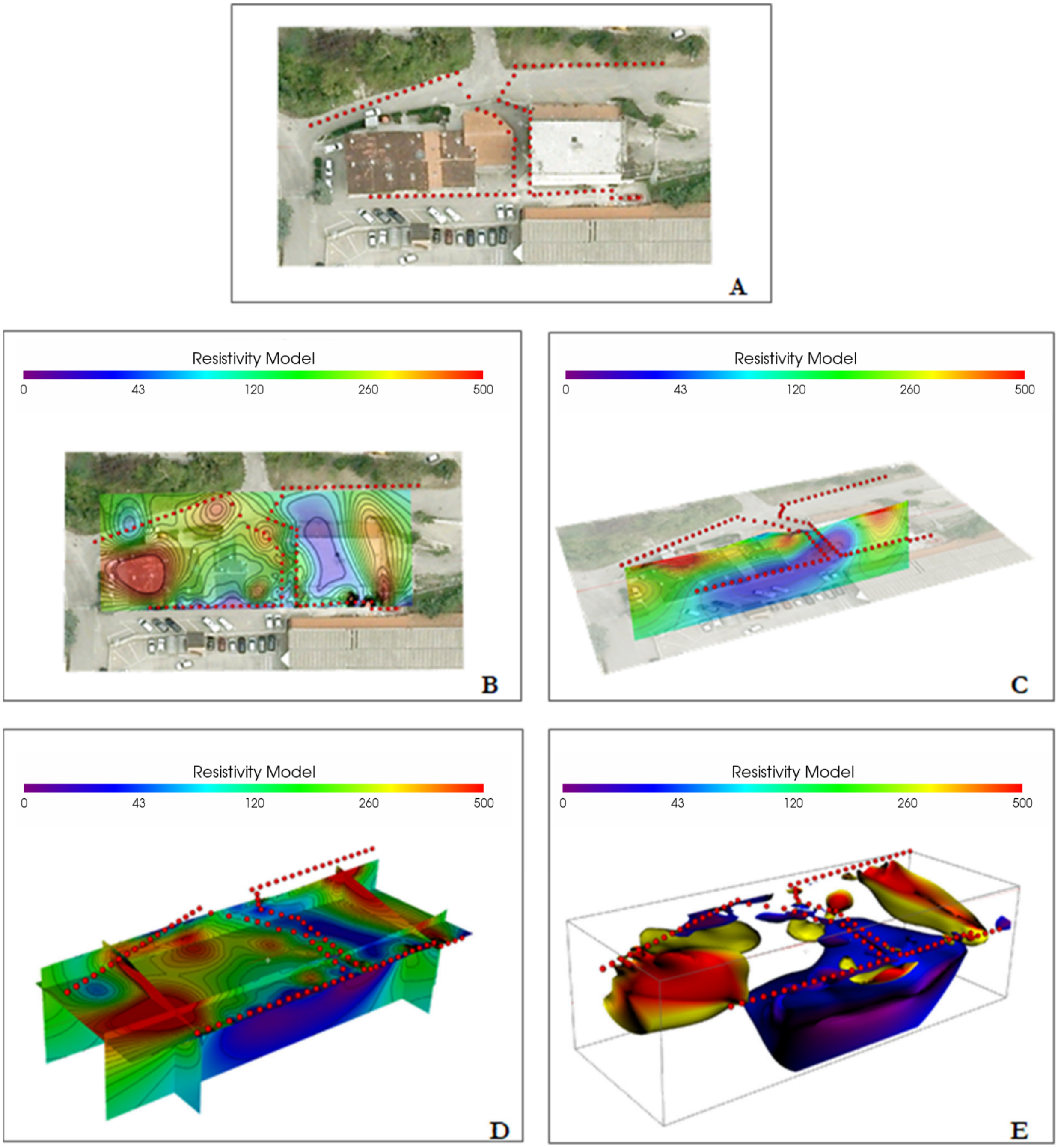


Figure 13.12: Some of many resistivity model project visualization possibilities. In A, aerial photo of the investigated site (from Google Earth) overlaid to electrodes used for the acquisition; in B visualization of Resistivity Model through the insertion of a XY section; in C insertion of a YZ section; in D insertion of 5 sections, 2 parallel to the YZ plane, 2 to XZ plane and 1 to XY plane; in E visualization of the mesh (outline only) and two volumes, one incorporating the most conductive areas (5-15 ohm*m, in blue) and the other the most resistive area (80-150 ohm*m, in yellow-red)

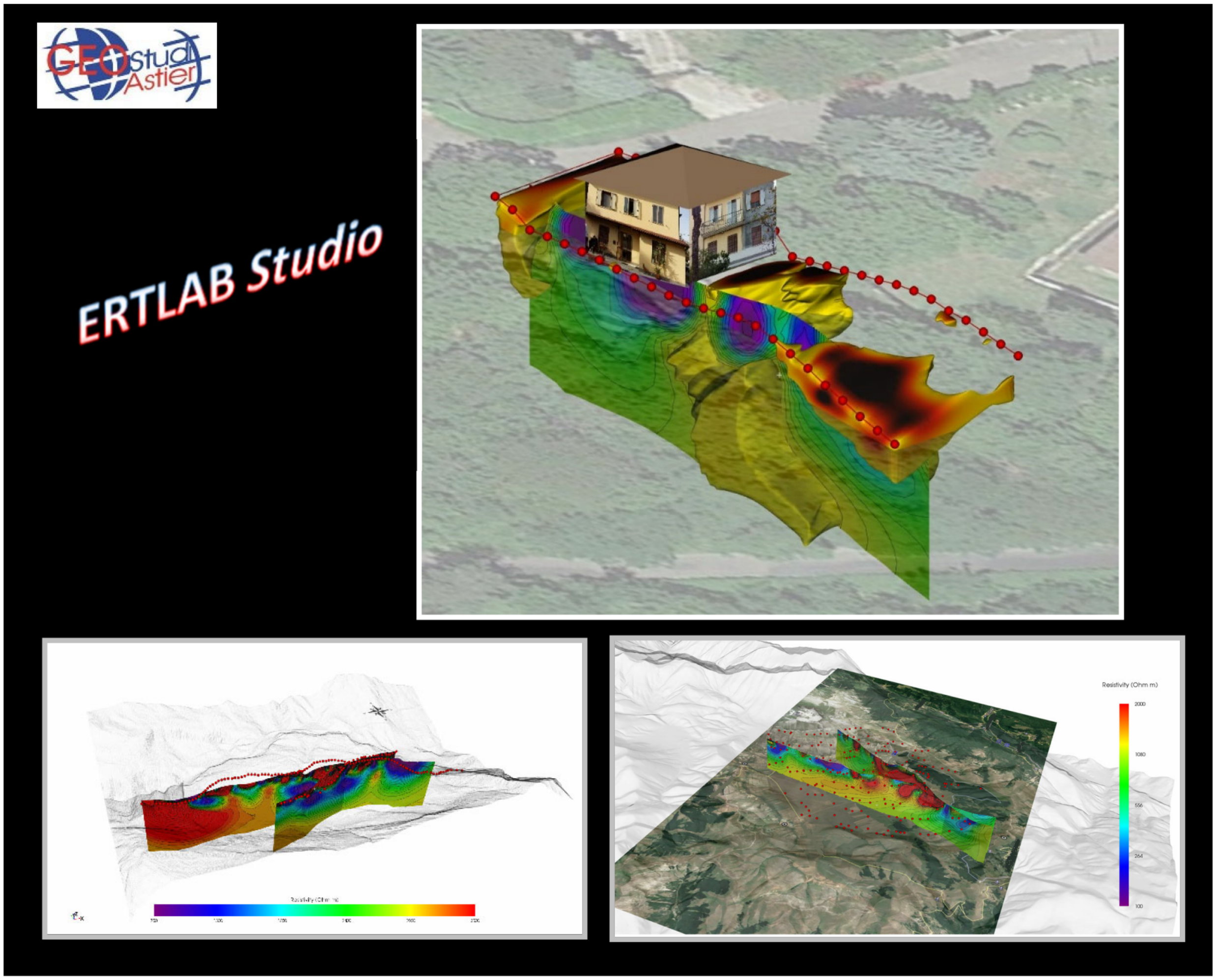


Figure 13.13: At the top, example of resistivity sections with aerial photo , electrodes, resistivity volume and custom building; on the bottom left, example of resistivity sections with electrodes and topography. On the bottom right example of resistivity section with electrodes, topography and aerial photo.

13.4 Conductivity Model

The *Conductivity Model* quantifies how easily the material allows the flow of electric current. It is the reciprocal of the resistivity model. As illustrated in Figure 13.14 the areas that are showing the maximum values in the conductivity model show minimum values in the conductivity model and vice versa.

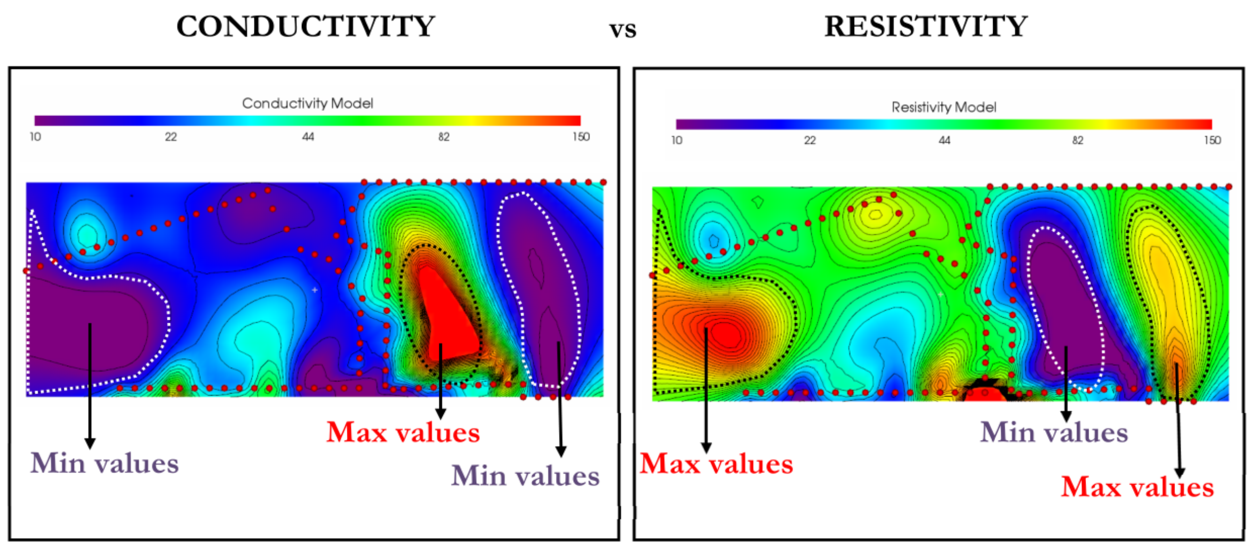


Figure 13.14: Comparison between Conductivity and Resistivity Models.

All its options are the same as those of the *Resistivity Model* section.

13.5 IP Model

In the IP methodology the ground is energized with an alternating square wave pulse and it consists in measuring the IP effect as a time-diminishing voltage at the receiver electrodes. The method is based on the observation of potential curve decay subsequent to the interruption of the input current, that is the extent of the "residual chargeability" retained by the soil under investigation. All its sub menus and options are the same of the *Resistivity Model* section.

13.6 Metal Factor Model

Warning: Feature not available in the basic license; included in the "Graphics pack" add-on.

This model is computed automatically based on the Resistivity and the chargeability (IP) models currently available. It is obtained dividing IP by Resistivity, and then a multiplier factor 1000 is applied to get more readable values. All its sub menus and options are the same of the *Resistivity Model* section.

13.7 Sensitivity Model

The sensitivity model indicates how much the variations of resistivity of subsurface will influence the potential measured by the electrode array. It provides an estimate of how much the electrical property of the ground must vary to produce significant effects on the measurement. Different arrays have different sensitivity patterns, some of them describe the superficial levels better others the deeper levels. To compute the sensitivity of the electrode configuration of the loaded project (just one quadrupoles or all quadrupoles together) use the Run Sensitivity option. In Figure 13.15 the area in red shows the maximum sensitivity, areas where collected data will be more reliable. The areas in purple are not optimally covered, as expected as the area with maximum sensitivity correspond to the area where the electrodes (yellow spheres in Figure 13.15) are positioned.

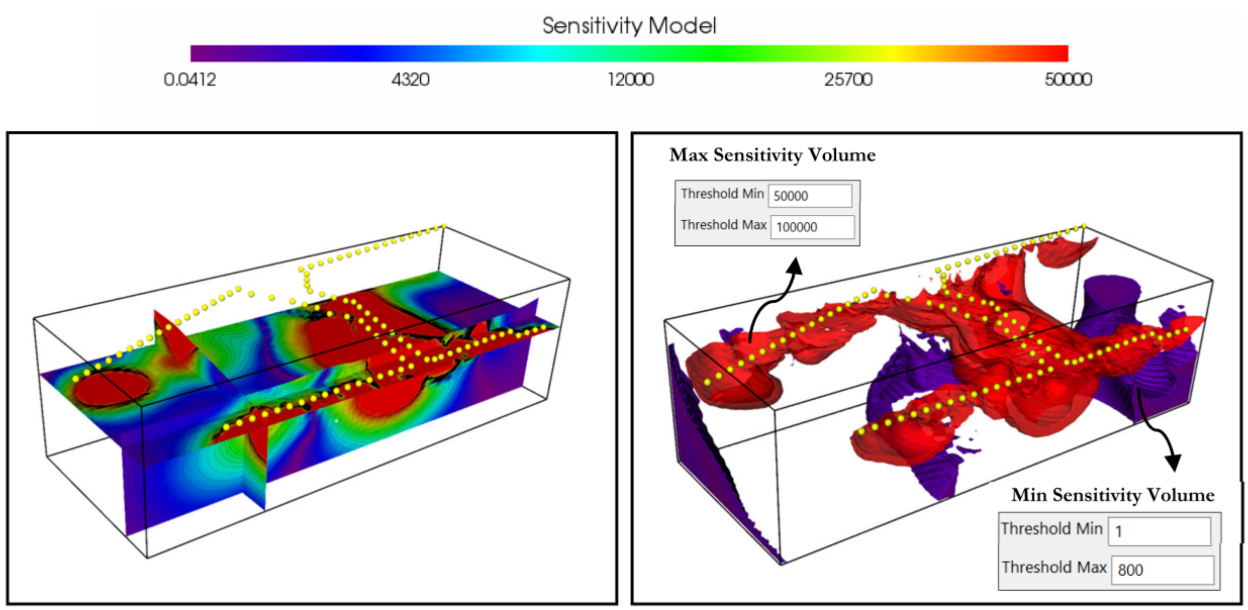


Figure 13.15: Sections and volumes of Sensitivity Model

13.8 Delimiter

Warning: Feature not available in the basic license; included in the "Graphics pack" add-on.

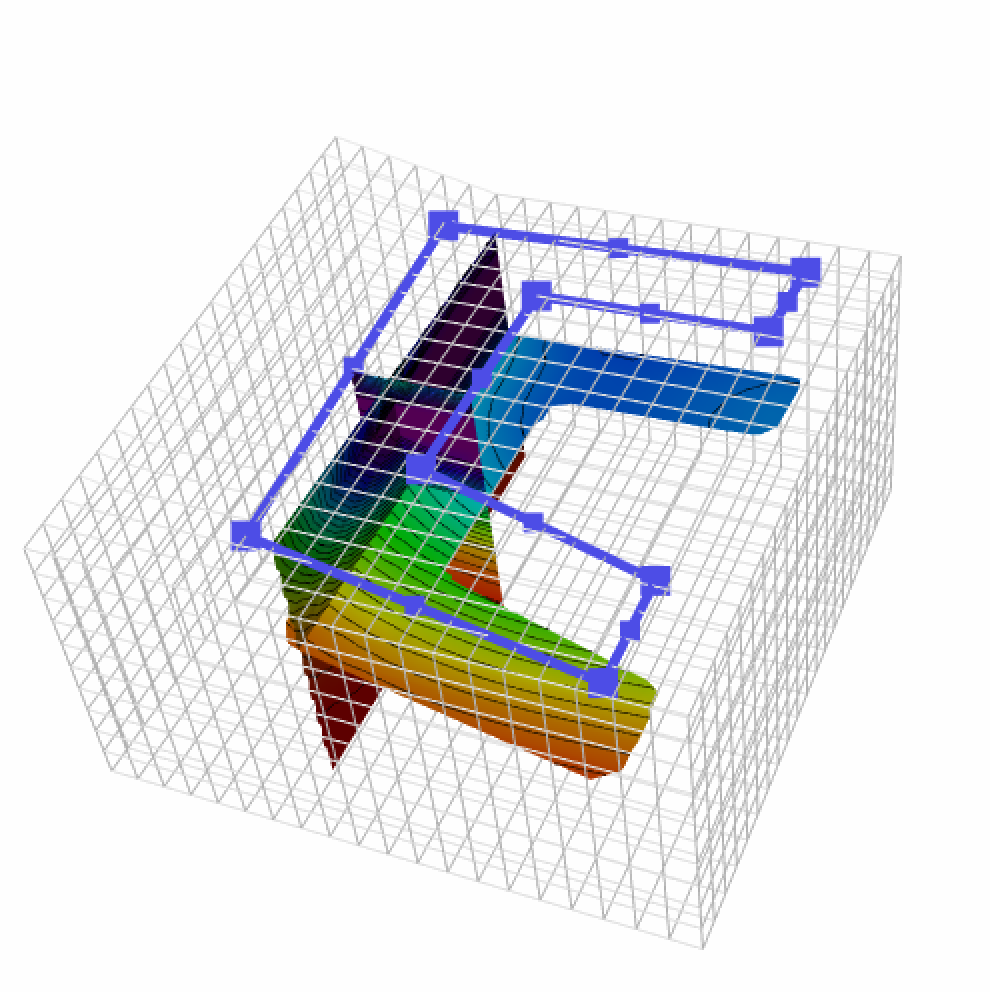


Figure 13.16: Delimiter

With this tool it is possible to show only the most interesting part of the mesh. To define how the mesh needs to be resized a polyline has to be defined.

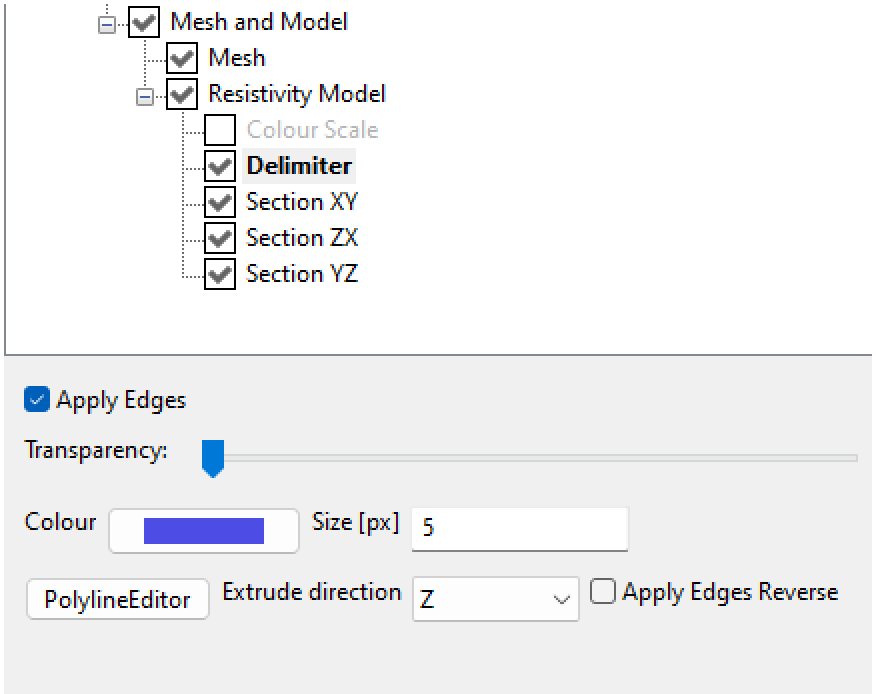


Figure 13.17: Delimiter parameters

It is possible to set the *Transparency*, *Colour* and the *Size* (in pixel) of the polyline provided. However the polyline is shown only when the *Polyline Editor* is open.

13.8.1 Apply Edges

Controls if the *Delimiter* is enabled, that means that it needs to effectively cut the mesh or not.

13.8.2 Polyline Editor

Same tool described here.

The only difference is that in this case the polyline is closed, that means that the first and last point provided are connected with an additional segment.

13.8.3 Extrude direction

Sets the direction (in the 3D space) where the polyline needs to be projected. The options are:

- **X**: The polyline is extruded horizontally, along X axis.
- **Y**: The polyline is extruded horizontally, along Y axis.
- **Z**: The polyline is extruded vertically, along Z axis (default).
- **Automatic**: The polyline is automatically extruded perpendicular to the mean plane on which it lies.

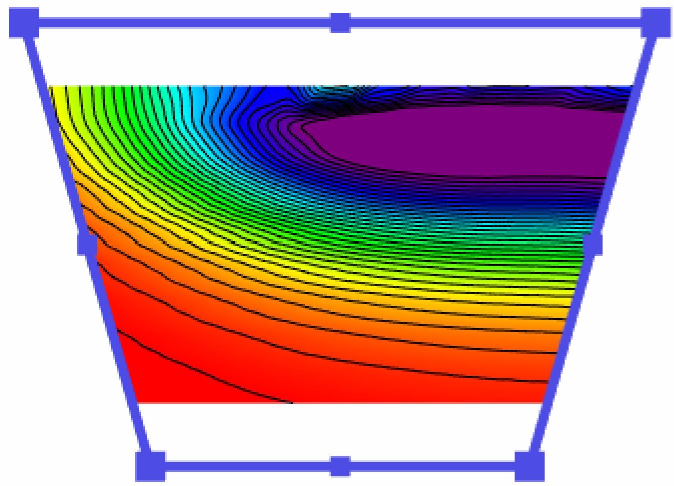


Figure 13.18: Delimiter example with horizontal extrusion to remove the corners to a 2D section

13.8.4 Apply Edge Reverse

To control which part (inner or outer) of the mesh to be shown with respect the polyline provided.

In Figure 13.15 is shown:

- **A** Delimiter disabled, shown the original mesh
- **B** Delimiter active, with *Apply Edge Reverse* disabled
- **C** Delimiter active, with *Apply Edge Reverse* enabled

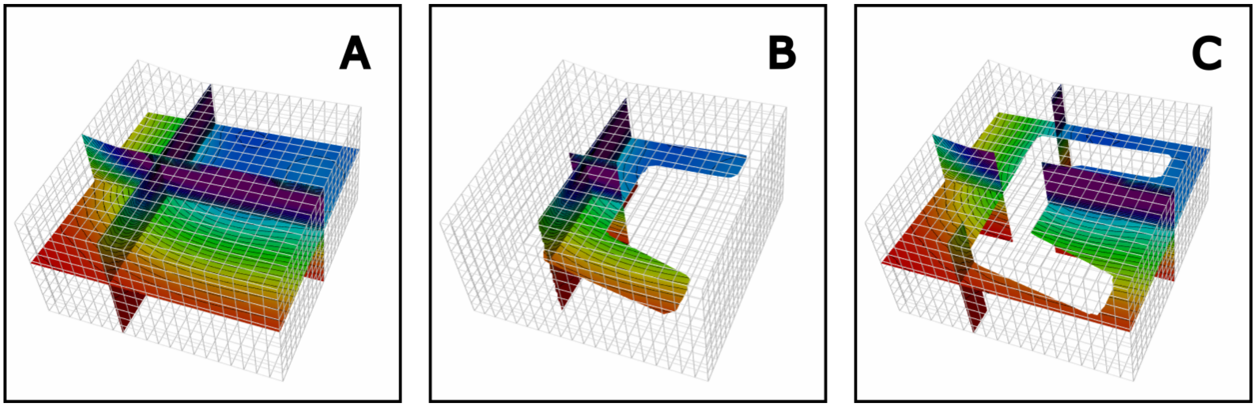


Figure 13.19: Delimiter examples

CHAPTER

FOURTEEN

MODEL SECTION

The *Section* tabs have the panel divided into: *Section* and *Contour*.

14.1 Section panel

The **Section** tab contains the following items:

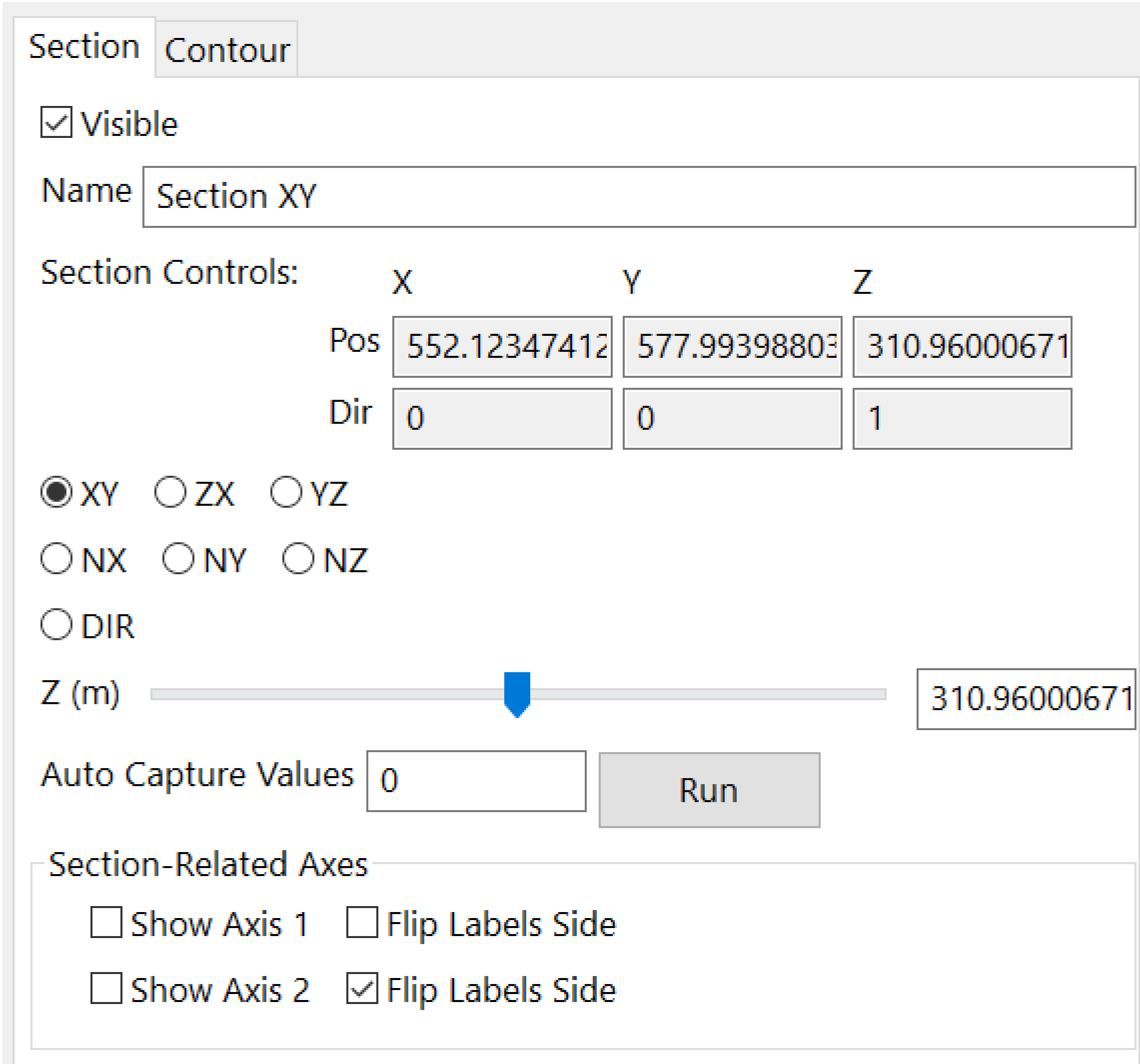


Figure 14.1: Section panel Tab

14.1.1 Visible

Controls the visibility of the Section;

14.1.2 Name

By default the node name is "Section XY/ZX/YZ", but it can be changed by entering the desired new name in this box;

Is a not an editable table. It shows the actual information about the *Position* and the *Direction* of the object in the 3D scene. It is automatically updated every time the operator clicks on the directional buttons explained below.

14.1.3 XY, XZ, YZ buttons

They allow to set the plane on which to visualize the section, choosing between these three main directions of the 3D space.

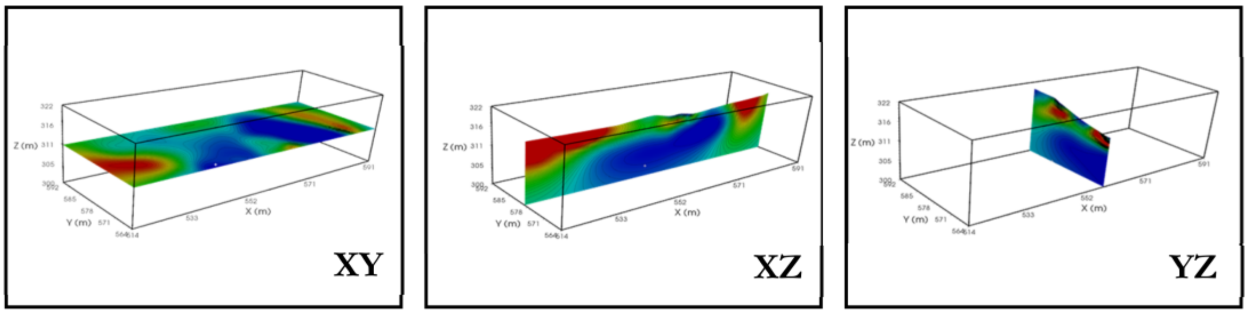


Figure 14.2: Section in the three main planes

14.1.4 NX, NY, NZ buttons

They allow to rotate the section by an angle *N* around the X/Y/Z axis. In the following example, the XY Section is tilted 20° around the X axis in one case and 10° around the Y axis in the other.

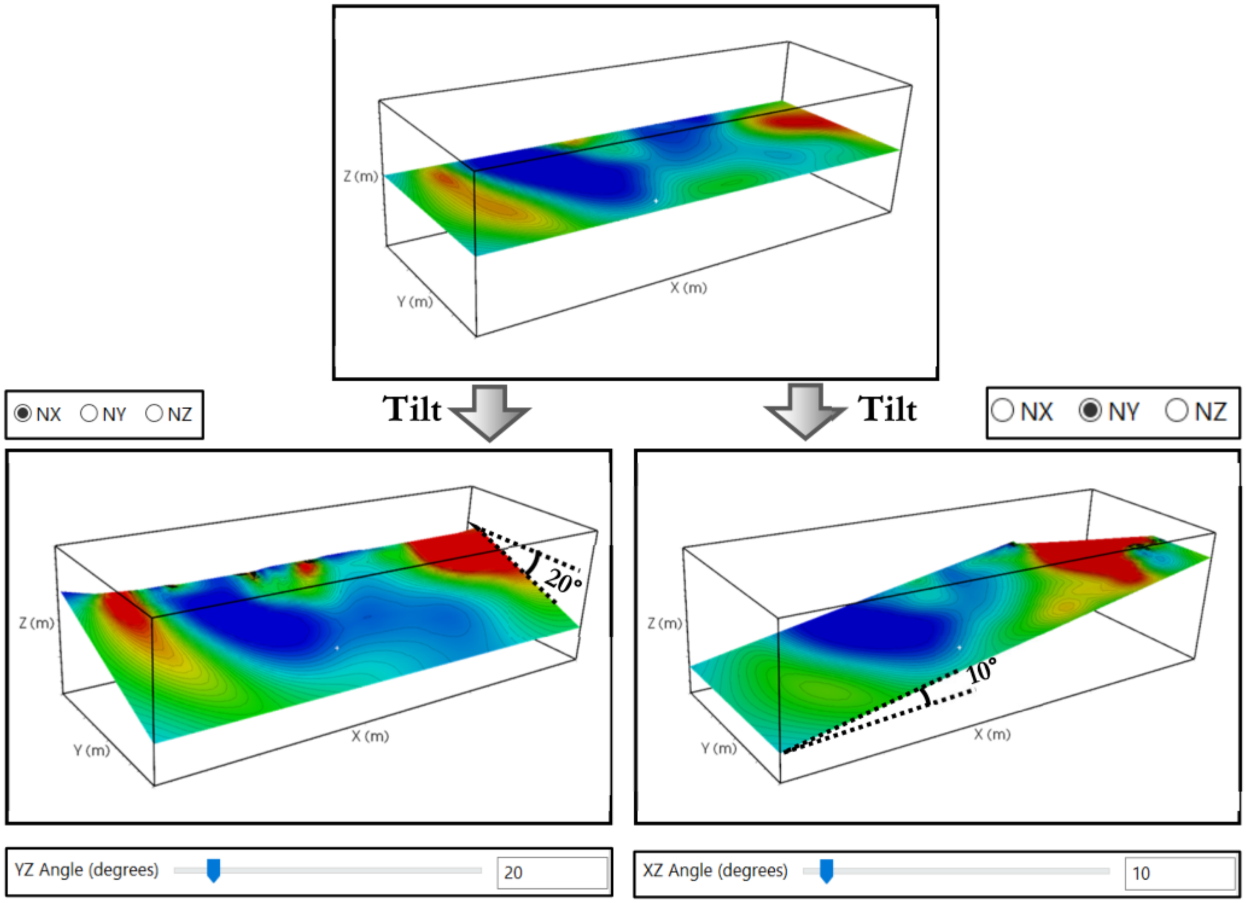


Figure 14.3: Examples of rotation of XY section

Note: Note that in some cases, during the mesh generation, a rotation can help to minimize the total number of cells. Then, to restoring the original coordinates, an opposite rotation is usually performed. After this it happens that the default sections, parallel to the main axes, can give questionable results. Because this it can be useful to set the *NZ* option, and to add/substract the rotation amount to the default *NZ* value, to then get a section that follows the local mesh axis.

14.1.5 DIR button

Allows to move sections in one of the main directions (checking NX, NY or NZ) in the mesh. Moving the slider, or typing the desired number in the number box, will move the section in the mesh until the desired position is reached. Some examples are shown in Figure 14.4.

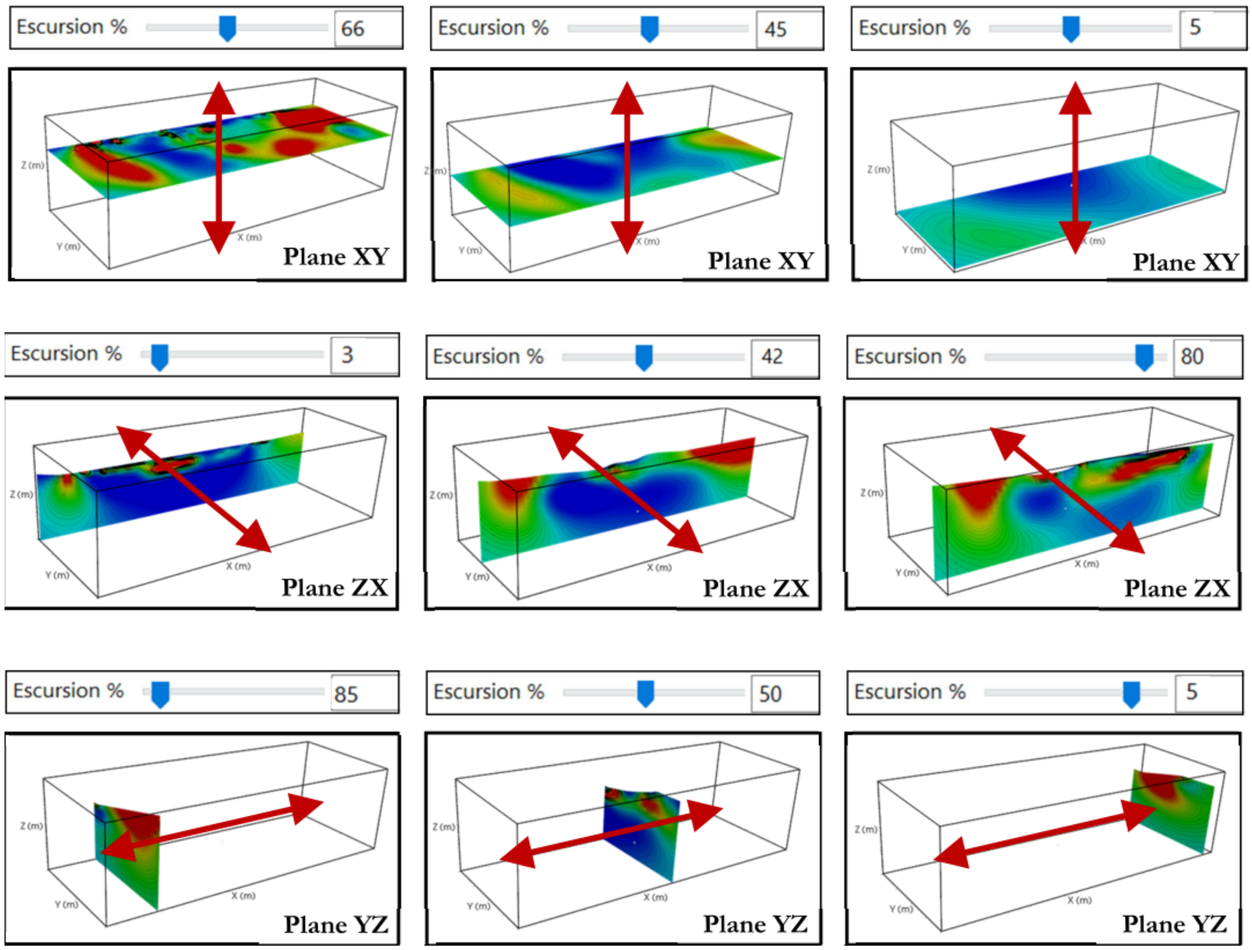


Figure 14.4: Examples of section translation through the three main directions

14.1.6 Auto Capture Values

Is useful when it is required to capture a lot of screenshots of the sections in the 3D scene from the same point of view of but in different positions. Type the sequence of numbers (one for each position) or using the specific notation (see section Mathematical Notations), as shown in Figure 14.5.

The numbers to type in the box refers to the label shown before the slider, it is X (m) in this case but it can be an angle (degree) if NX, NY, or NZ is selected.

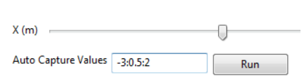


Figure 14.5: Auto-capture values panel

In this case a series of images will be saved with the sections from positions -3 to position 2, along the X axis, with a step size of 0.5m (11 images throughout).

Clicking on the *Run* buttons automatically saves the images in the working folder. The same result can be obtained using the "*Capture*" button at the top of the screen in the 3D scene, but in this case it is necessary to do save every figure individually. Using the "*Auto Capture Function*" allows to save many images all at once.

14.1.7 Section-Related Axes

This option allows to add a second axis system, applied to the selected section. Check *Show Axis 1* to visualize the vertical axis and/or *Show Axis 2* to visualize the horizontal axis. If the labels are not readable because they are superimposed on the Section (Figure 14.6) they can be flipped to the opposite side of the axis line using the *Flip Labels box*.

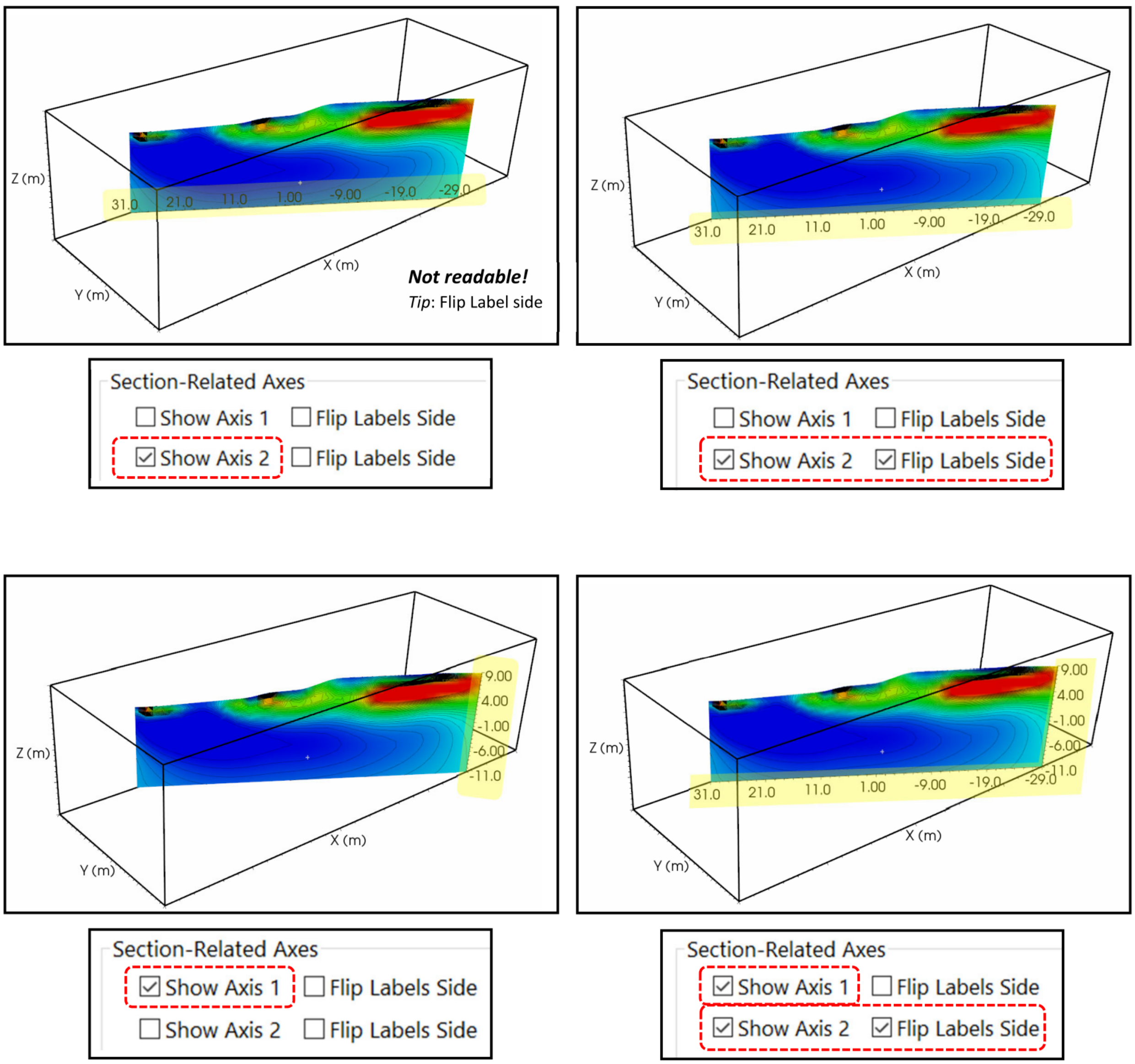


Figure 14.6: Examples of secondary axes visualization

14.2 Contour Panel

The **Contour** tab contains the following items:

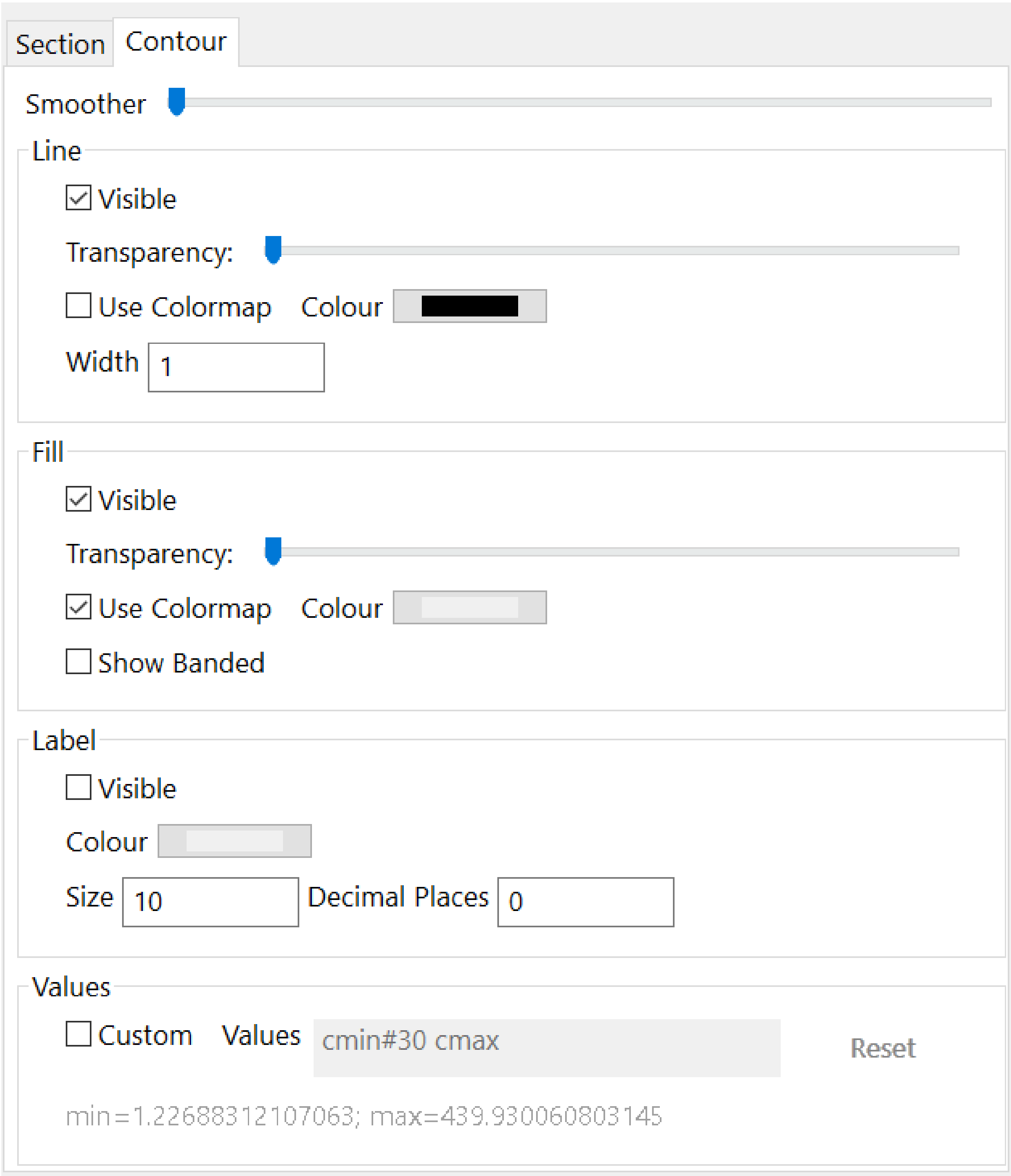


Figure 14.7: Contour Tab

It is divided into 4 sub-panels: *Line*, *Fill*, *Label*, and *Values*.

14.2.1 Line

The *Line* panel has the following options:

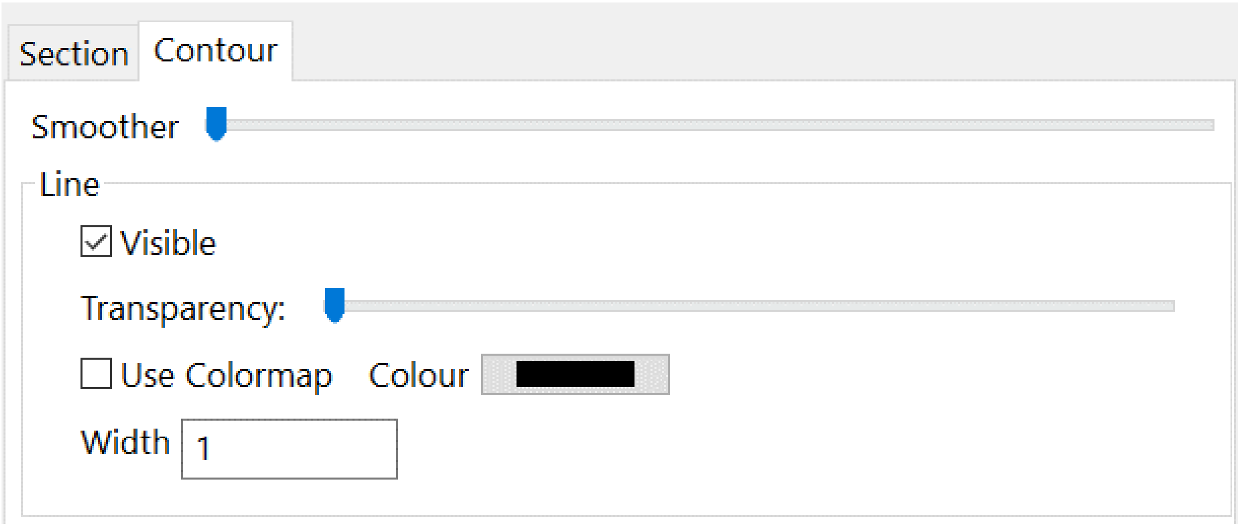


Figure 14.8: Contour Lines setting

- **Visible**: controls the visibility of the contour lines (Figure 14.9).

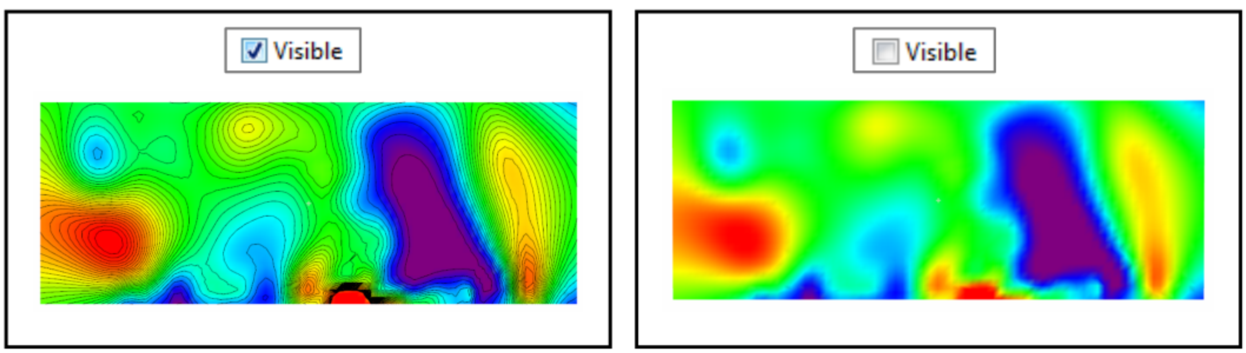


Figure 14.9: Contour lines visible and not on a XY section

- **Transparency**: manages the transparency level of the lines. When the slider is totally to the left the lines are completely visible, when it is on the right they are not visible and at the intermediate positions they are more transparent as the slider goes towards the right.

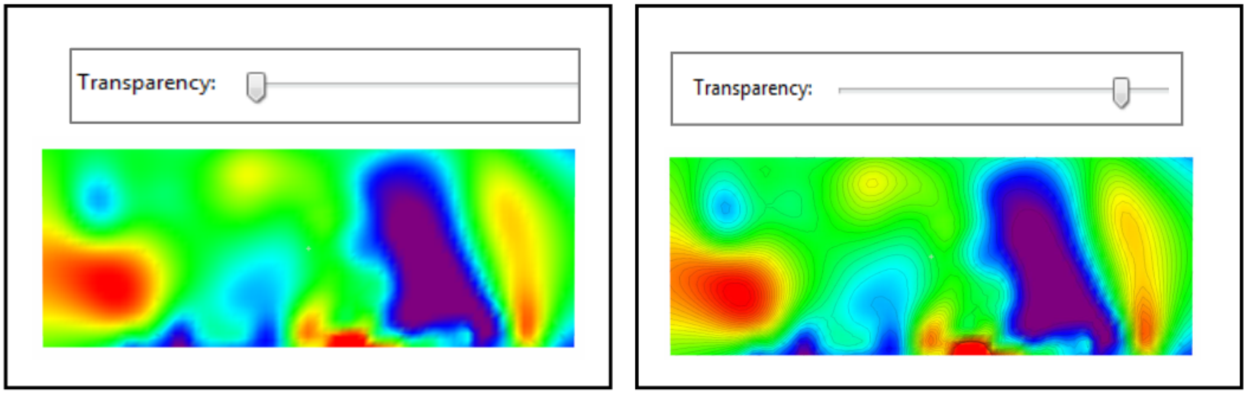


Figure 14.10: Different level of contour lines transparency

- **Use Colormap/Colour**: if checked the contour lines have the colour of the Colormap. Their colour changes as function of the values. If it is not checked, all the lines have the same colour, set by the apposite button (it is dark grey by default).

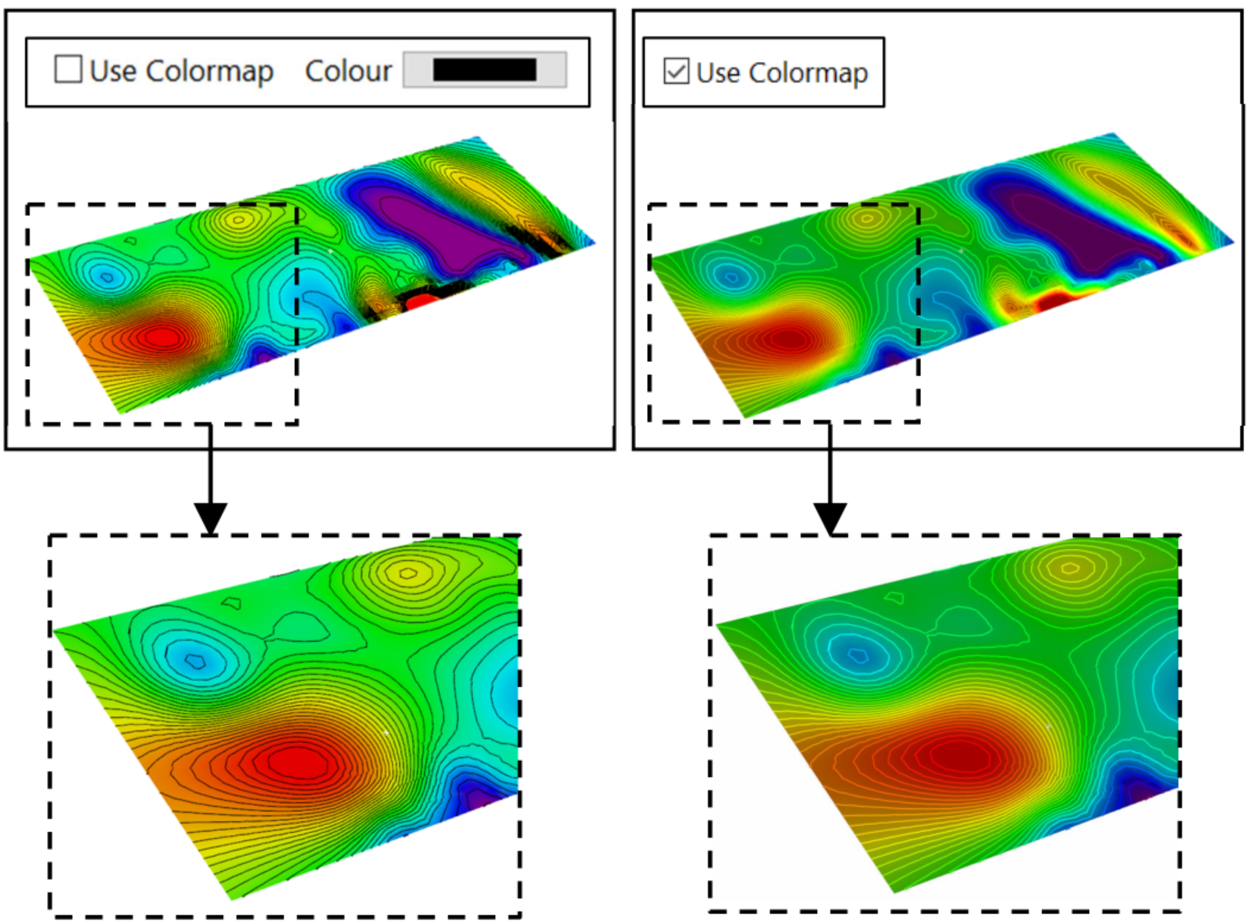


Figure 14.11: Example of one colour (black) and colormap contour lines

- **Width**: set the width of the colour lines, typing a number from 1 to 20 in the box.

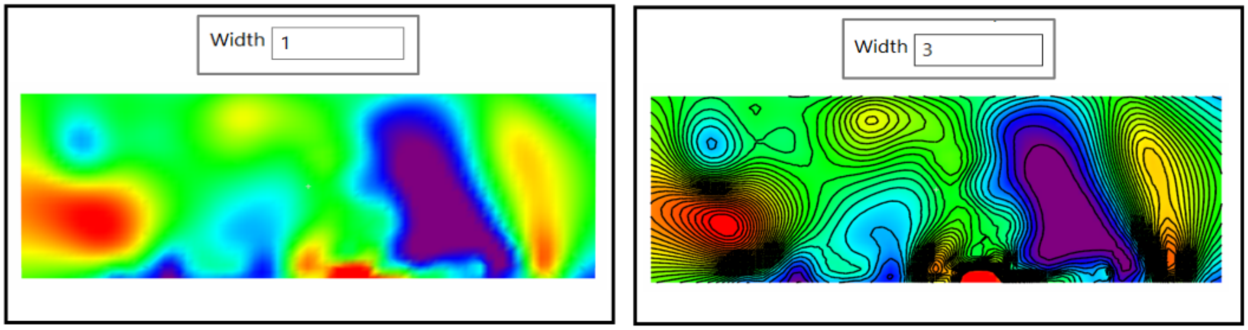


Figure 14.12: Example of different contour lines width

- **Smoother**: smooths the appearance of the contour lines in case they are not round shape enough.

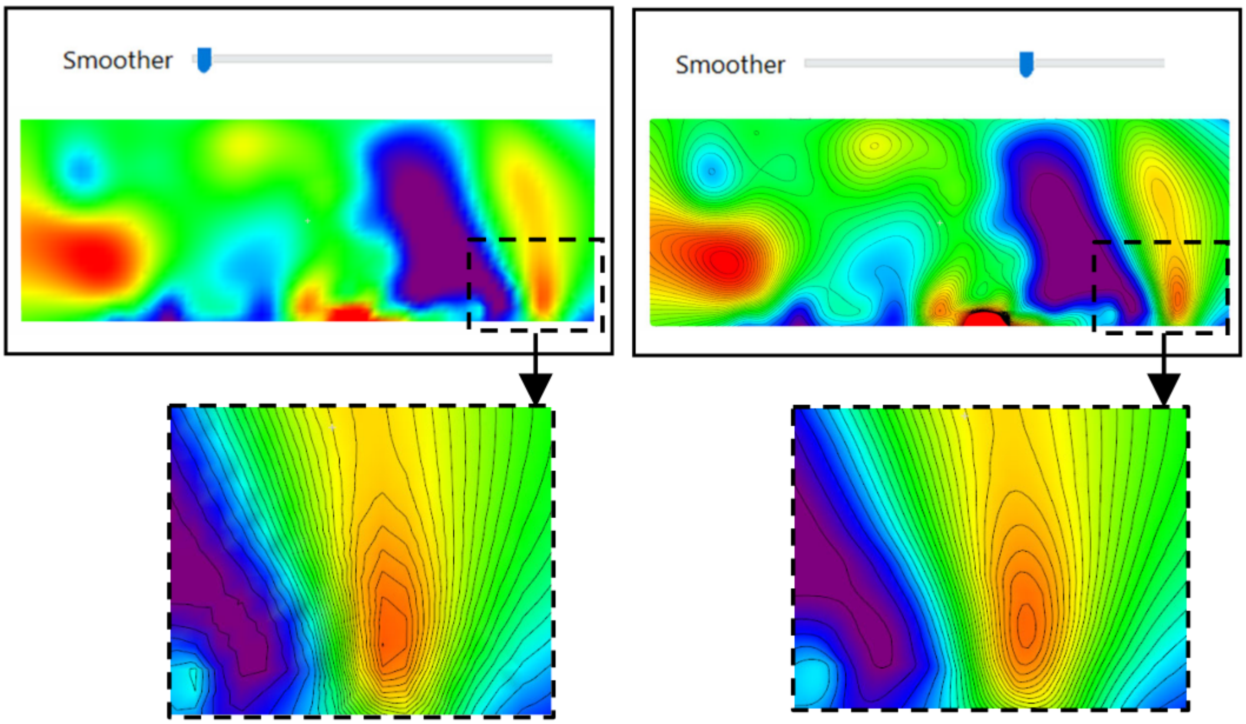


Figure 14.13: Example of contour lines smoothing

14.2.2 Fill

The *Fill* panel has the following options:

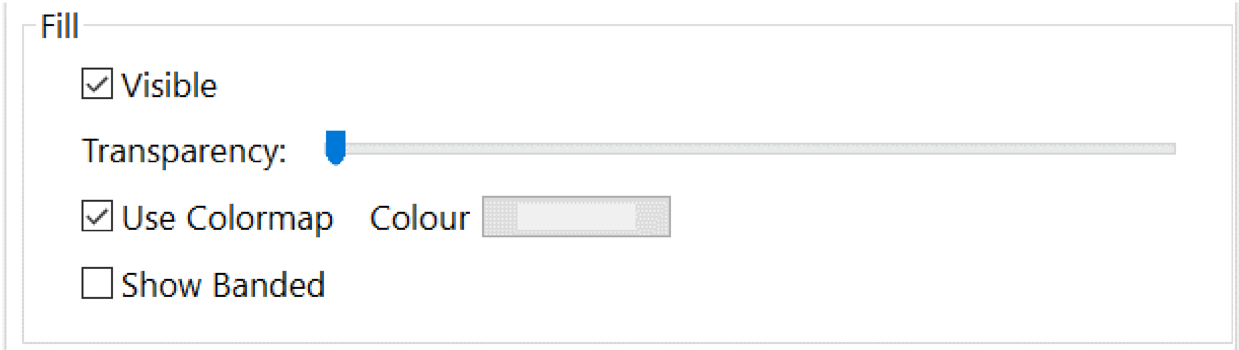


Figure 14.14: Contour Fill setting

- **Visible**: controls the visibility of the contour lines. They are normally coloured in every point. If *Visible Line* is selected, it displays the section with the contouring lines overlaid. If it is not checked, only the contouring lines are visible (Figure 14.15).

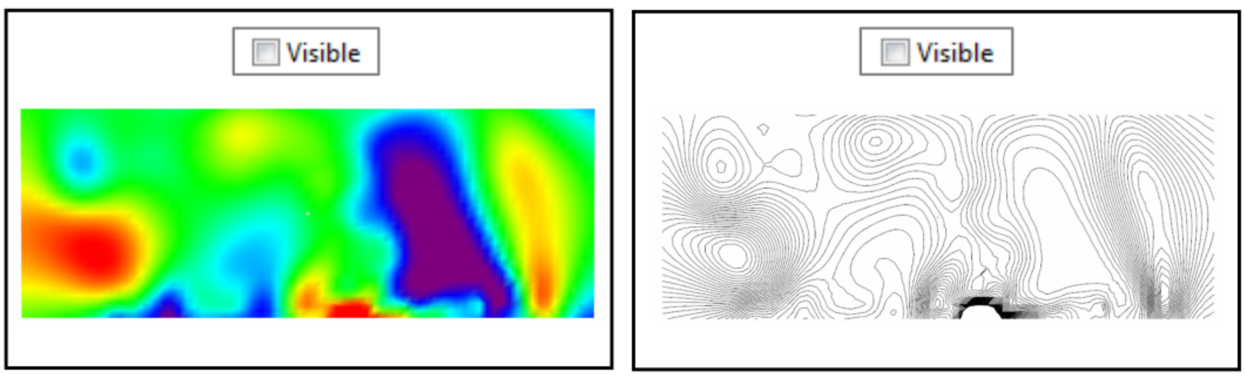


Figure 14.15: XY section with and without the filling visible

- **Transparency:** manages the transparency level of the filling (Figure 14.16).

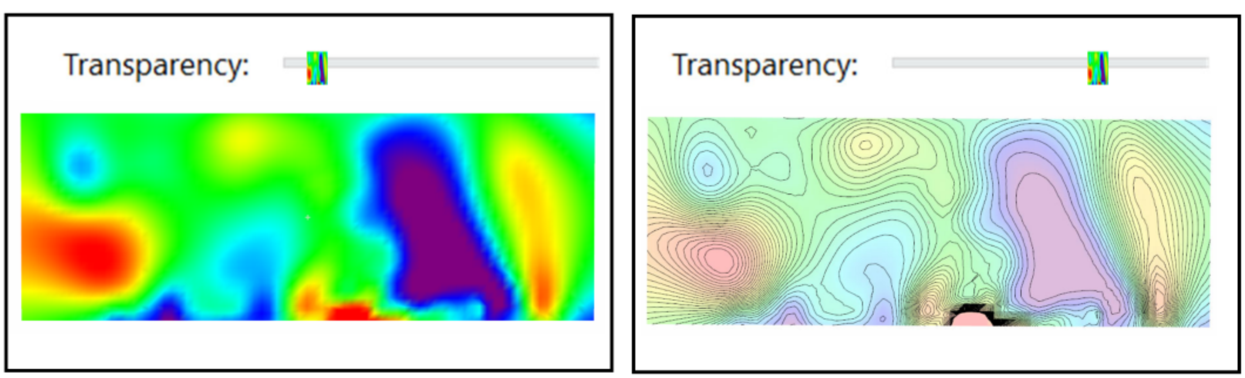


Figure 14.16: Different level of filling transparency

- **Use Colormap/Colour**: if checked the filling has the colour of the Colormap. It changes as function of the resistivity values. If not checked the area has the same colour, set by the button nearby (it is dark grey by default). In Figure 14.17 on the right the contour lines follow the colourmap.

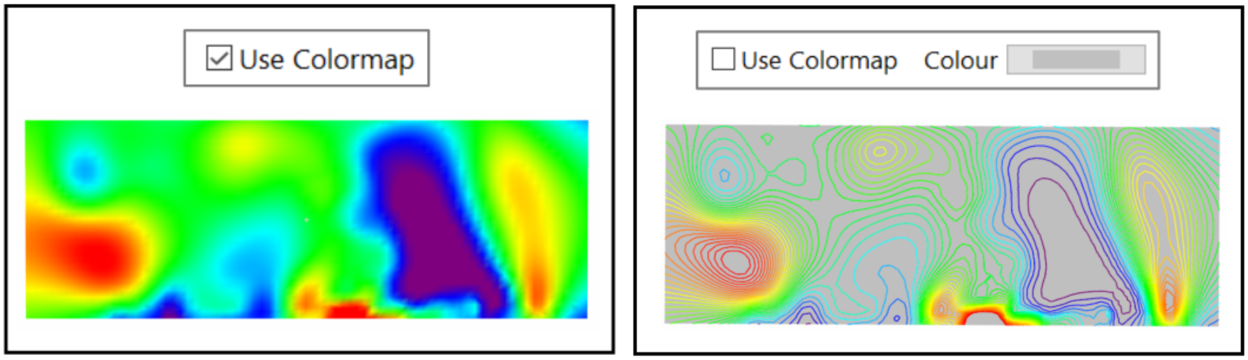


Figure 14.17: Example of one colour (grey) and colormap filling

- **Show Banded**: if checked the filling is banded when it goes from one value to another. If unchecked the transition from one colour to another is nuanced. The difference between the two ways to represent data is more evident if the contour lines are not visible.

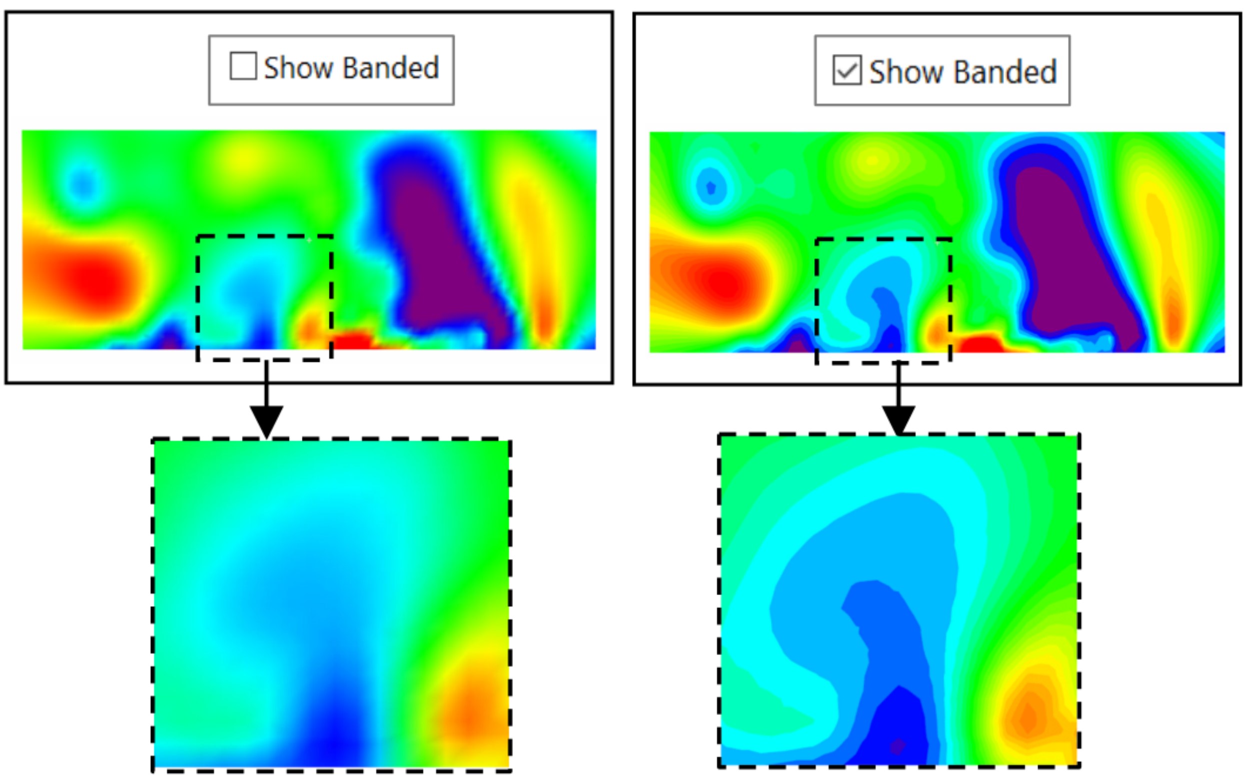


Figure 14.18: Filling of XY section with banded visualization mode active and not

14.2.3 Label

The *Label* panel has the following options:

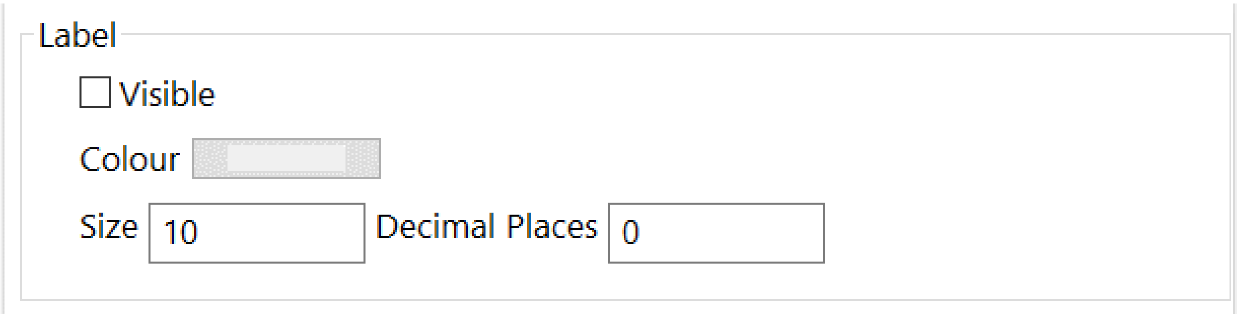


Figure 14.19: Contour Label setting

- **Visible**: controls the visibility of the contour lines labels.

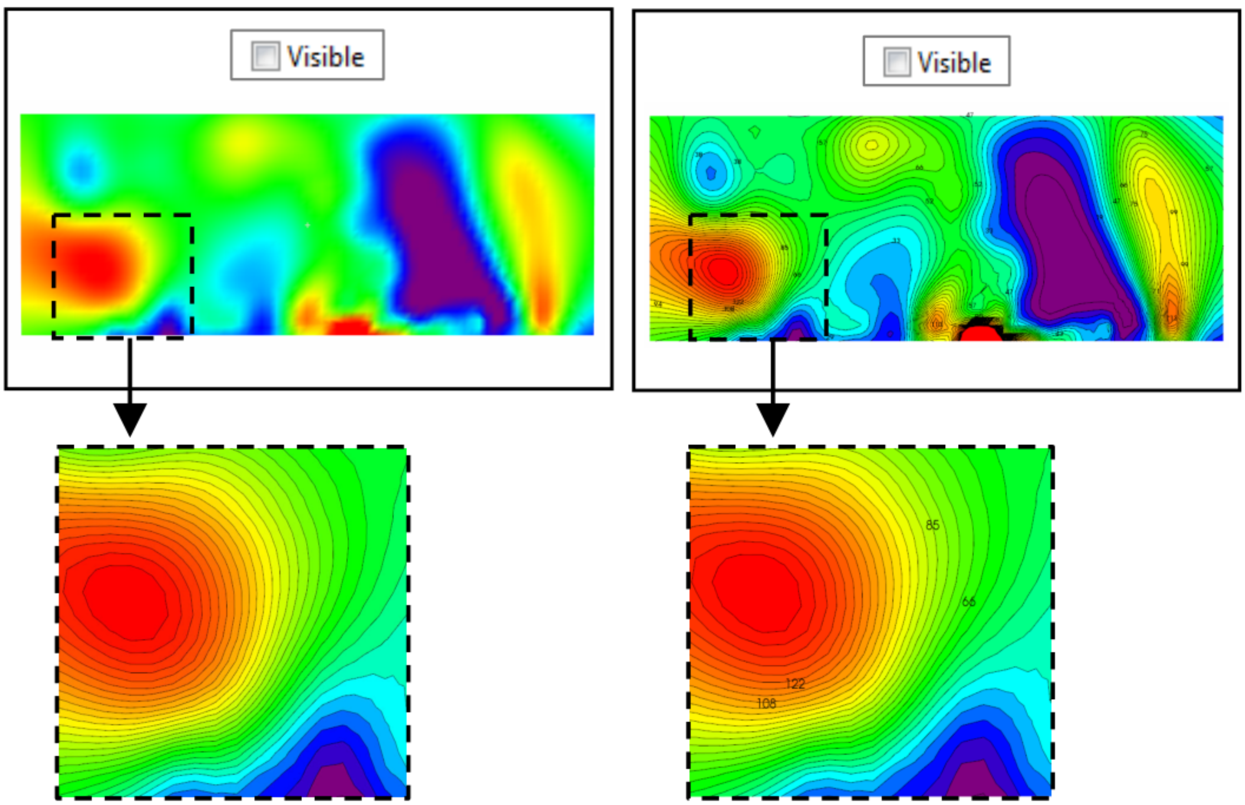


Figure 14.20: Filling of XY section with and without lines labels visible

- **Colour:** set the colour of the contour lines labels through. It is grey by default. In the following example it is changed from black to white.

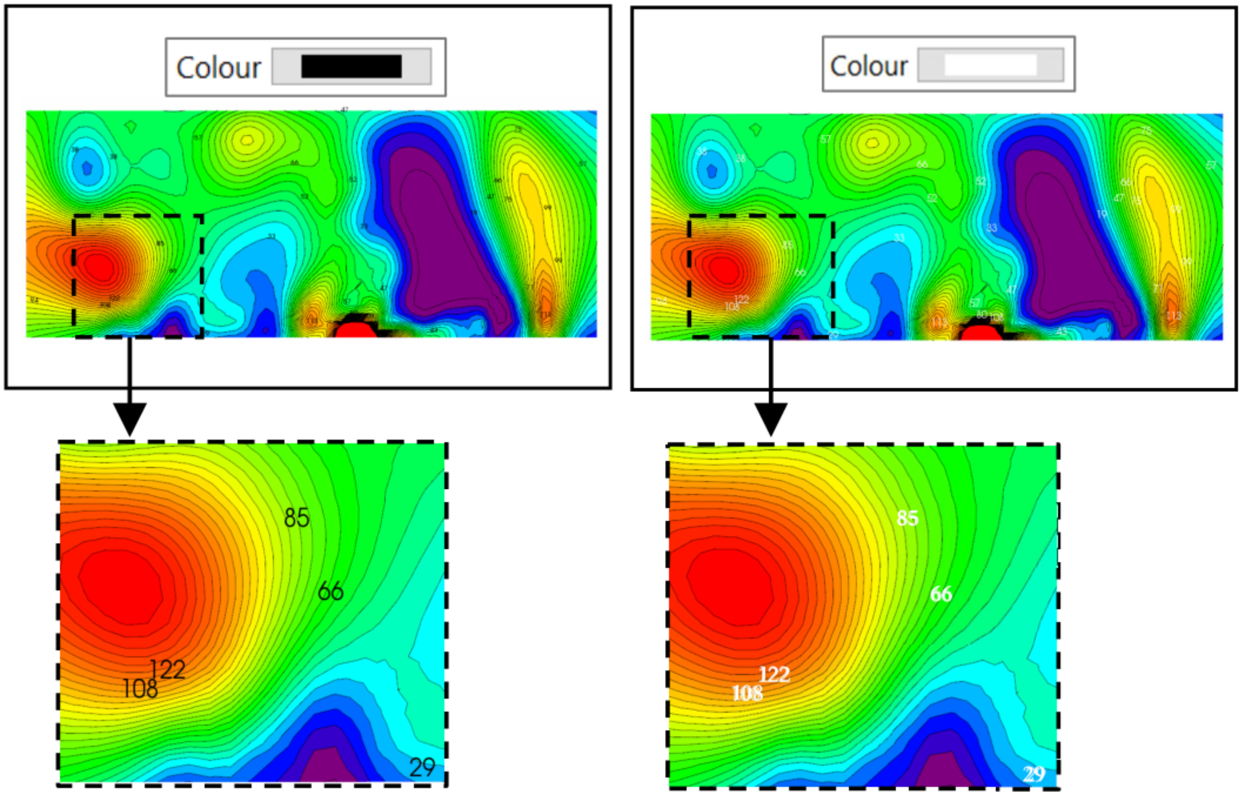


Figure 14.21: Example of lines labels of different colours

- **Size**: set the size of the contour lines labels, writing a number from 1 to 50 in the number box. In the following example it is set at 10 and 20.

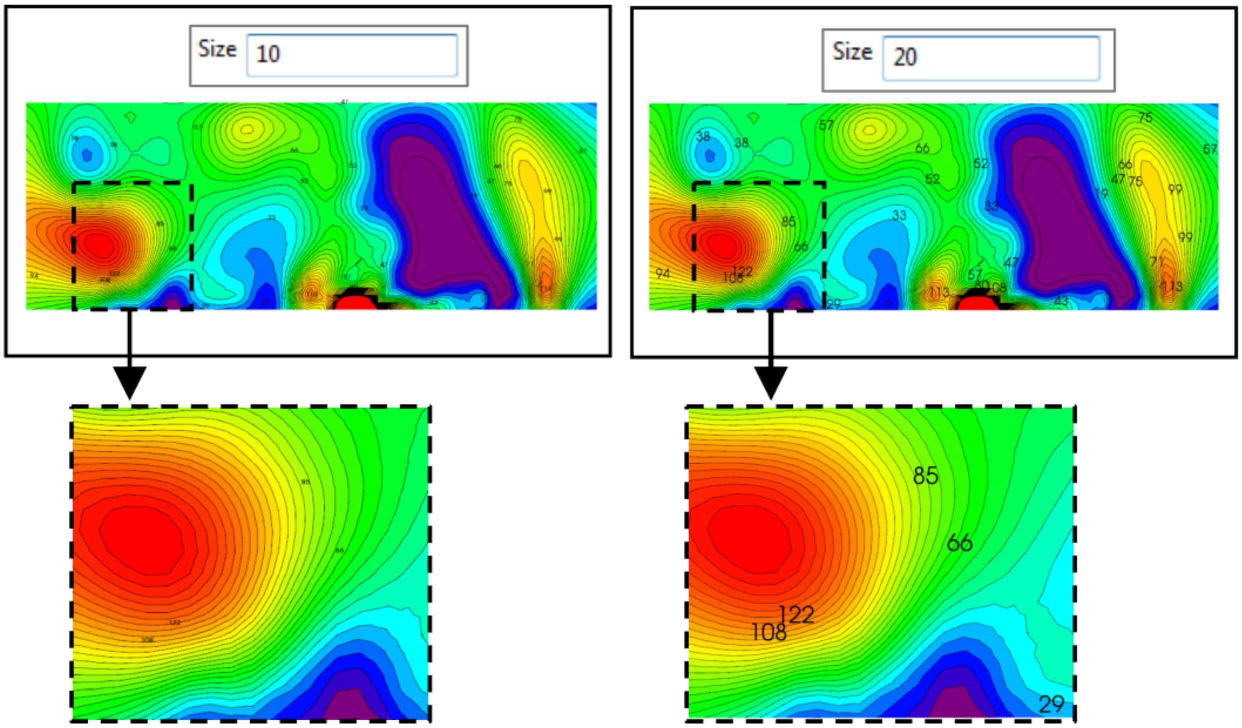


Figure 14.22: Example of lines labels of different sizes

- **Decimal Places**: set the number of contour lines decimals. It is set to 0 by default, but it can be changed by typing a number from 1 to 50 in the number box. In the following example it is set to 1 and 2.

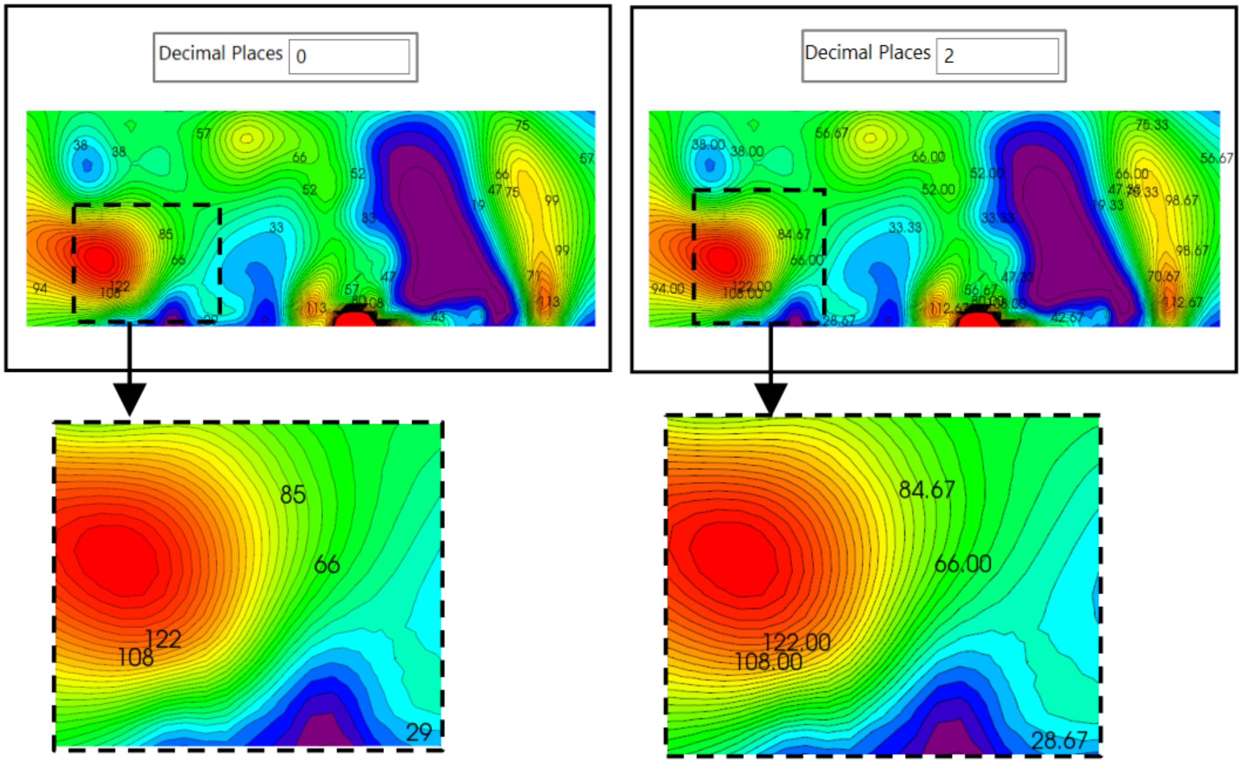


Figure 14.23: Example of lines labels with different decimal places

14.2.4 Values

The *Values* panel has the following options:

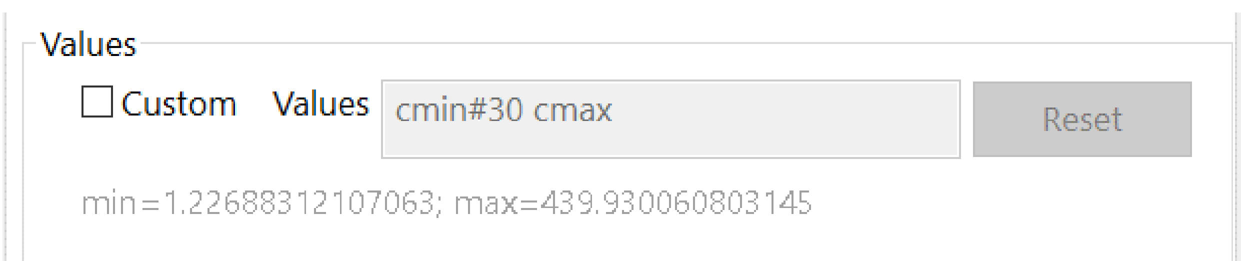


Figure 14.24: Contour Values setting

Here the maximum and minimum values to apply to the contouring lines and the step interval from one line and the next can be selected. Below the text box the minimum and maximum values of the entire dataset are reported.

- **Custom**: if not checked the optimal values are automatically calculated. If checked the text box is activated.

- **Values** box: type the desired number of contour lines to display in the text box. Type the sequence of numbers or use one of the mathematical notations described in Mathematical Notations. One of the correct ways to write the numbers is: *Minimum value to represent*: Step *:* Maximum value to represent

In Figure 14.25 values from 5 to 150, with step 10 and 20 have been chosen for visualization.

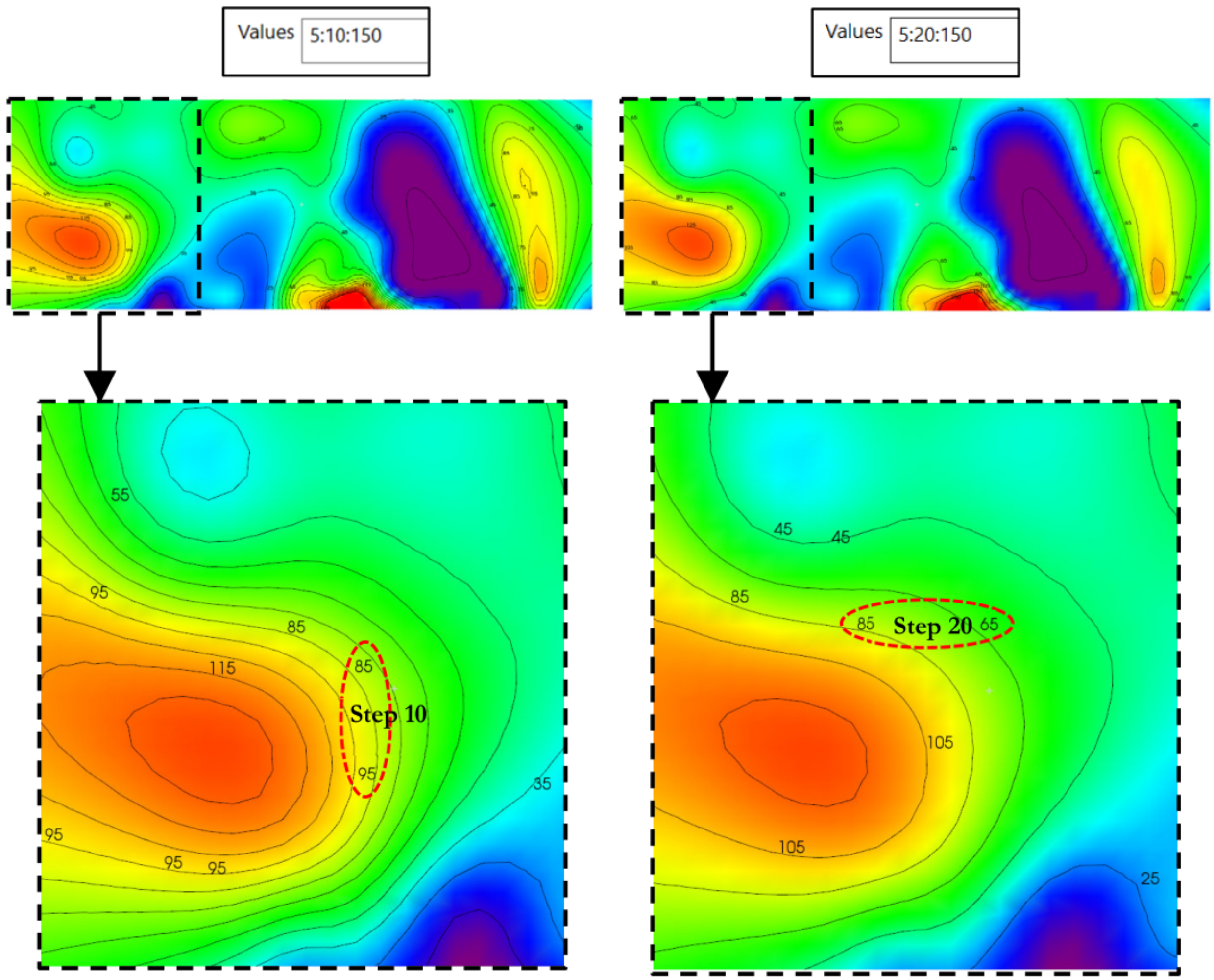


Figure 14.25: Example of different step value between contour lines

CHAPTER FIFTEEN

MODEL FENCE SECTION

> **Warning:** Feature not available in the basic license; included in the "Graphics pack" add-on.

The *Fence Section* is similar to the *Section* panel, but it is used to make cuts to the mesh that are not strictly planar. The main difference in this case is that a polyline needs to be defined instead of setting the cutting plane position and orientation.

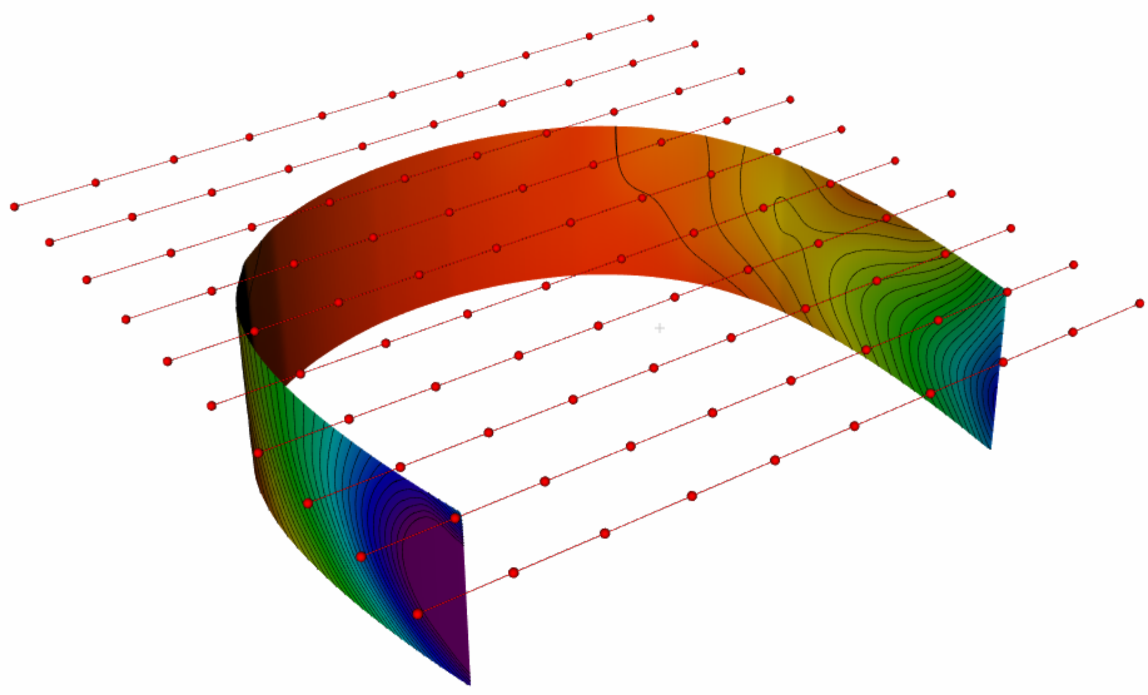


Figure 15.1: Fence Section

The *Fence Section* tabs have the panel divided into: *Section* and *Contour*.

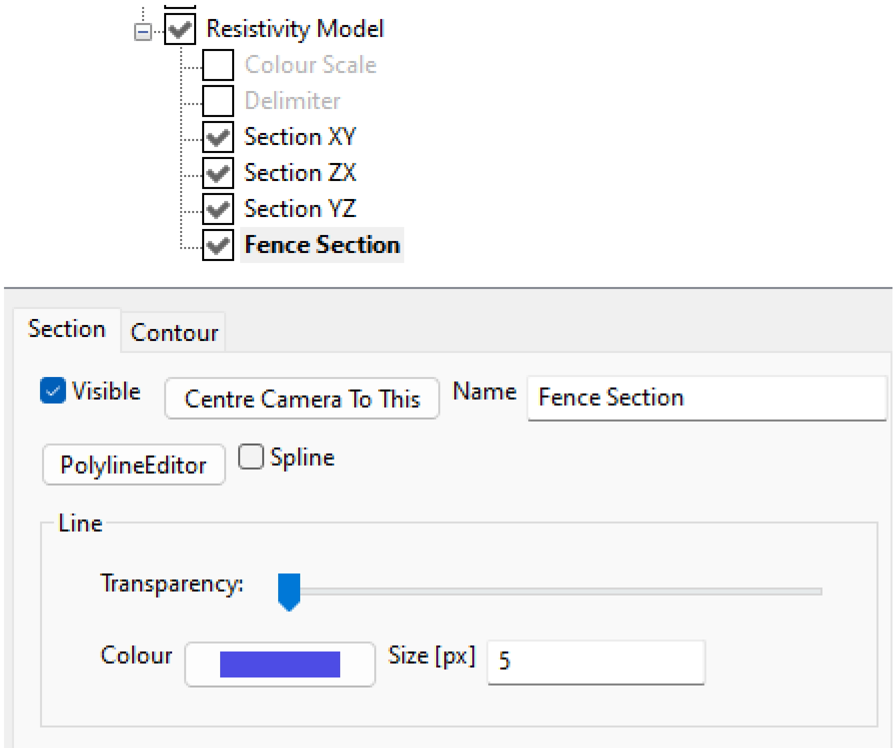


Figure 15.2: Fence Section properties

15.1 Section panel

The **Section** tab contains the following items:

15.1.1 Visible

Controls the visibility of the Section.

15.1.2 Centre Camera to This

To force the camera to view this object.

15.1.3 Name

To set the name of this object as desired.

15.1.4 Polyline Editor

Pressing this button it is opened a window to edit the polyline provided.

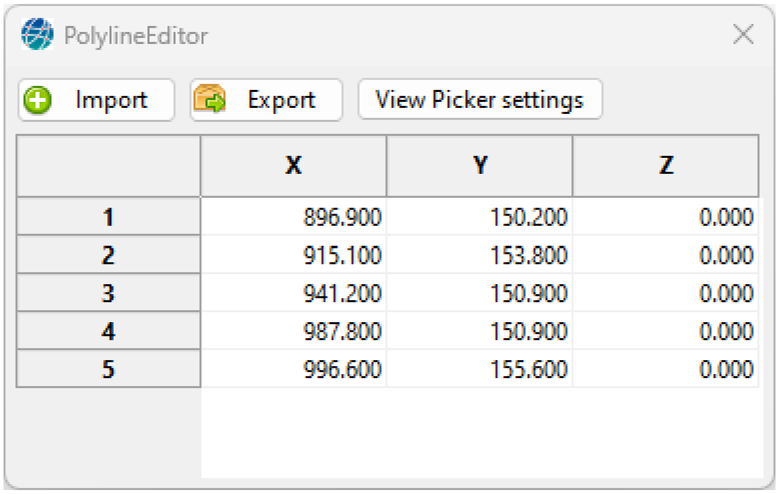


Figure 15.3: Polyline Editor

- *Import* To load a text file, with numbers in a 3 column format.
- *Export* To save a text file, with numbers in a 3 column format.
- *View Picker setting* To open the global View Picker settings tool.

Note: To place the polyline exactly under the electrodes it can be useful to Export the electrode positions to a file, then to use the *Import* button in this window to load it.

With the table it is possible to edit manually the coordinate list.

To add a new row at the end of the table then right click with the mouse the 3D scene, the cursor position will be taken, according to the picker settings.

To remove one (or more) rows from the table, select them, and press the *Delete* key from the keyboard.

The polyline can be also edited with the mouse, interacting with the 3D scene.

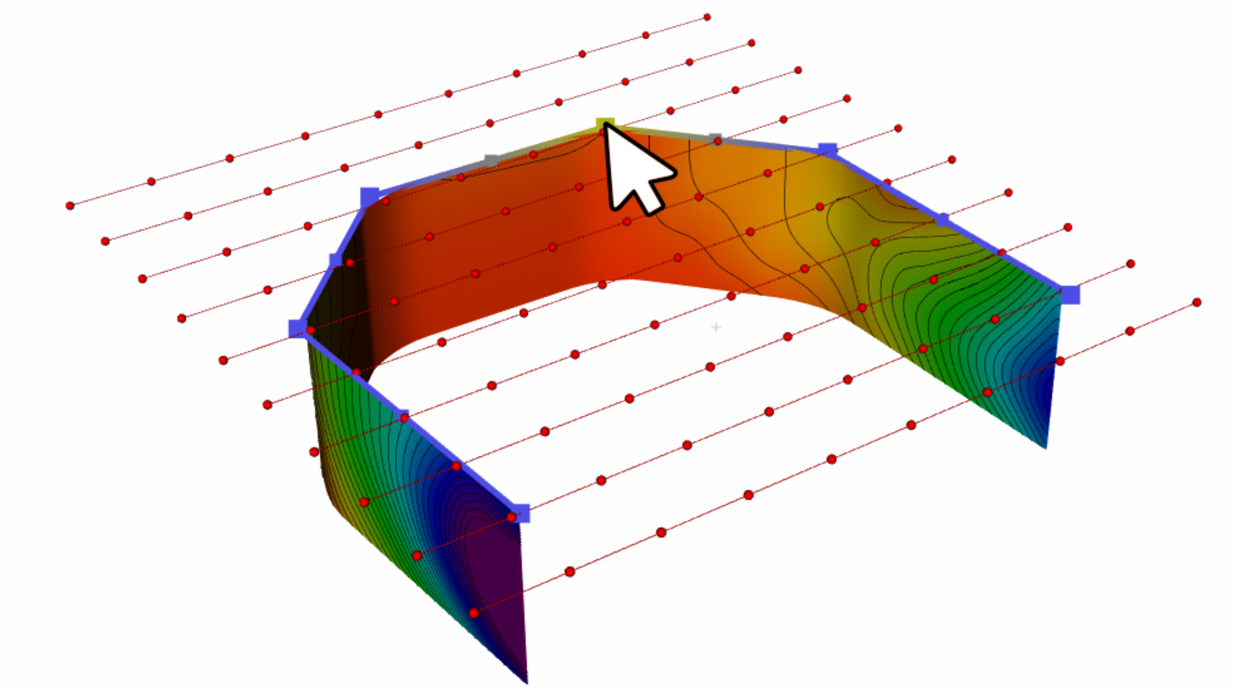


Figure 15.4: Polyline Editing

- *Edit a point*: move the mouse over the desired point, it should change colour, then press the right mouse button and drag and drop it in a different position.
- *Delete a point*: move the mouse over the desired point, it should change colour, then press the *Delete* key in the keyboard.
- *Insert a new point*: can be performed like the *Edit a point* feature, but it is then needed to select one of the *virtual* points that are drawn in the middle position between adjacent *real* points.

Note: Note that to correctly select a point in the 3D scene the picker needs to be correctly set to let the mouse cursor be relative the correct point in the 3D space. This is also true when trying to move a point in a different position.

15.1.5 Spline

To smooth the polyline as a continuos curve, or to draw the polyline as a sequence of linear segments.

15.1.6 Line

To set how the polyline is drawn in the scene. It is possible to set the *Transparency*, the *Colour* and the *Size* (in pixel) of the line.

15.2 Contour Panel

For this node see Model Section.

CHAPTER SIXTEEN

MODEL VOLUME

this menu option allows to identify the volume of data in the model with the same resistivity/conductivity/IP/sensitivity values, the iso-volume. The window has the following items:

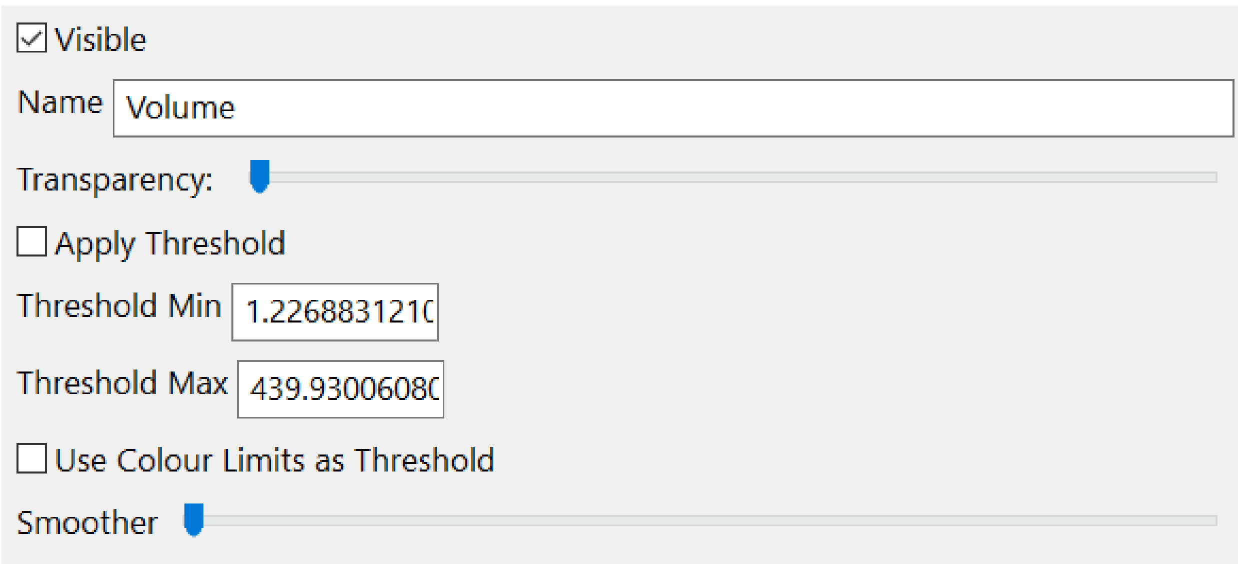


Figure 16.1: Volume panel

16.1 Visible

Sets the visibility of the *Volume*.

16.2 Name

Assign to the *Volume* (default name) the desired name by typing in the text box.

16.3 Transparency

Manages the transparency level of the volume. When the slider is totally at the left the object is completely visible. When it is on the right it is not visible at all and at intermediate positions it is more transparent as the slider goes towards the right. In the following example the volume of the entire dataset and a section are displayed. In the first case (Figure 16.2, case A) the transparency of the volume is zero and the section is not visible. In the second case (Figure 16.2, case B) the level of transparency is high and the vertical section appears.

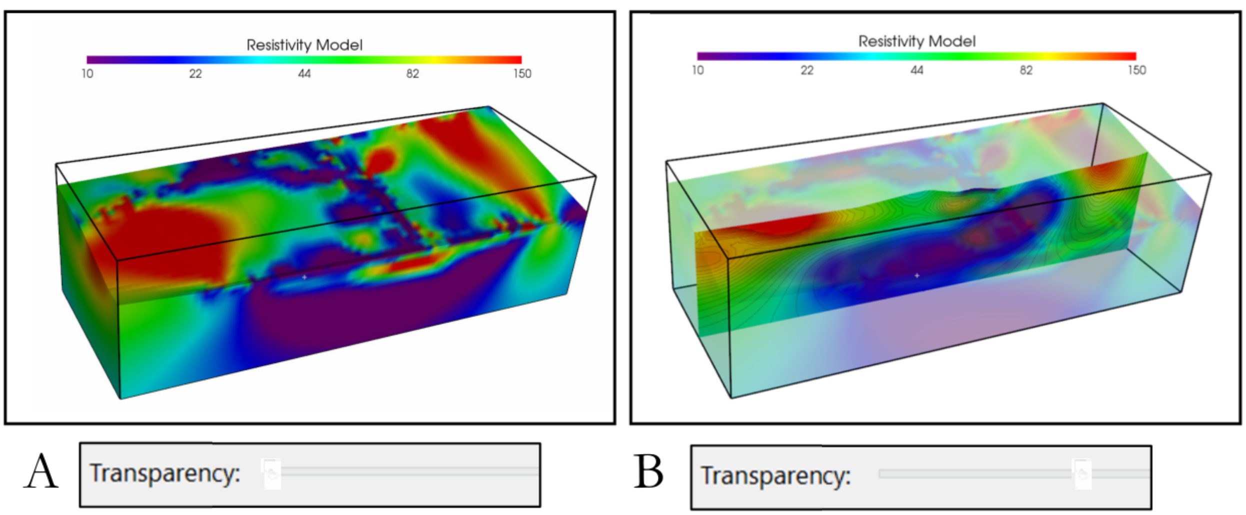


Figure 16.2: Different levels of transparency in the resistivity volume

16.4 Apply Threshold

To visualize a desired volume, type in the desired values in the *Threshold Min* and *Threshold Max* boxes and check *Apply Threshold* to activate the selection. If the box is not checked the entire range of data is shown. In the following example, a low resistivity model, in blue, and a high resistivity model, in red, are shown (Figure 16.3).

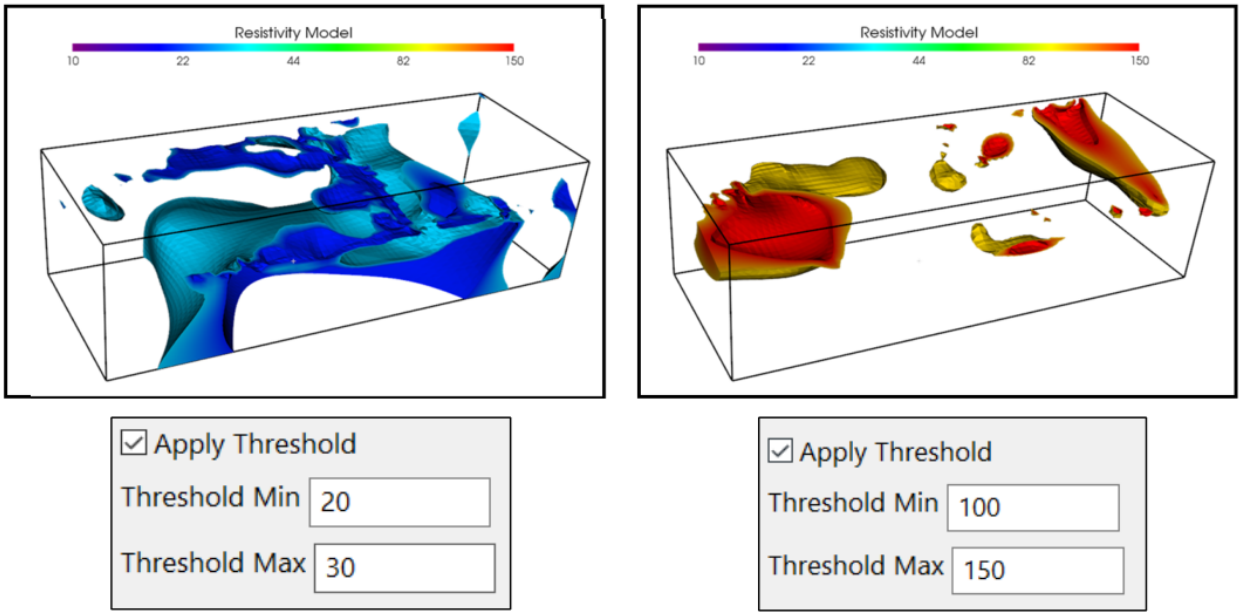


Figure 16.3: Example of two different volumes threshold applied for the creation of maximum and minimum resistivity volume

To visualize additional volumes at the same time add one *Volume* node for each sub-selected range values (Figure 16.4).

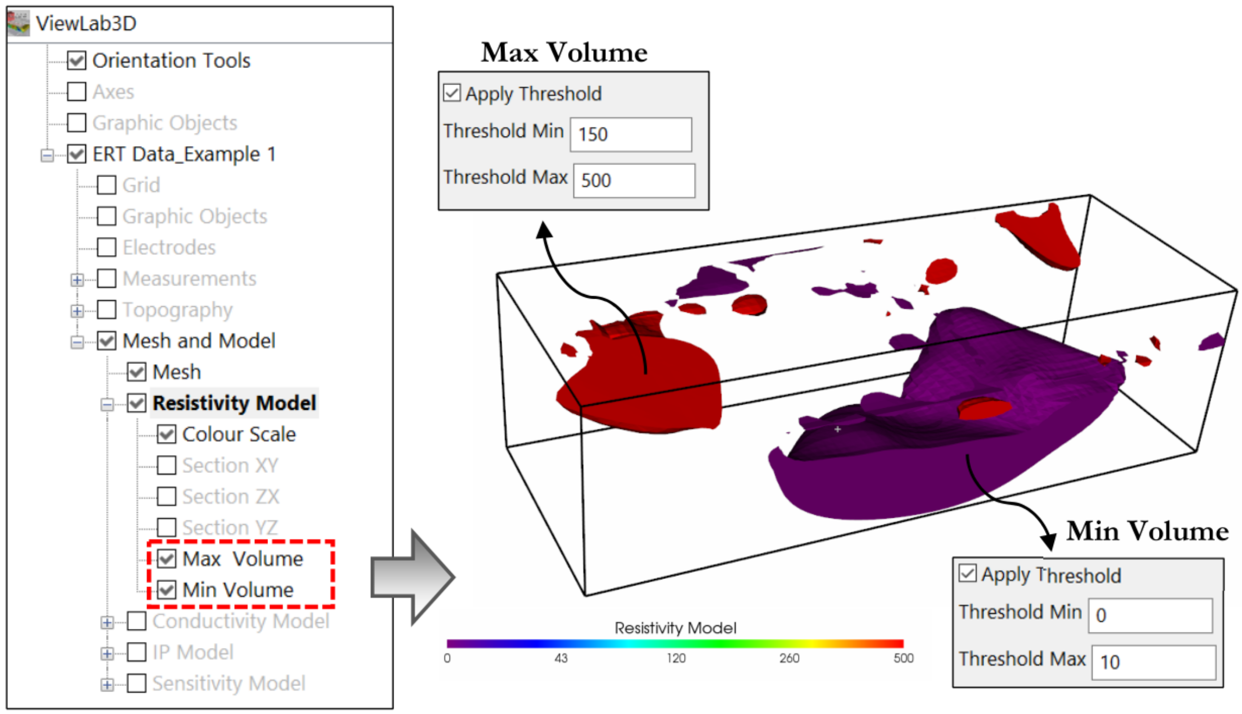


Figure 16.4: Example of visualization of maximum and minimum volume at the same time

16.5 Use colour Limits as Threshold

by default the *Threshold Min* and *Max* boxes are set to the minimum and maximum values of the entire dataset. To use the custom extreme values of the colour scale as maximum and minimum value of the volume, check the *Use colour Limits as Threshold* box (and *Apply Threshold* to make the selection valid). In this way the saturated area, external to the colour scale value, are cut away from the volume and a “hole” appears in their place.

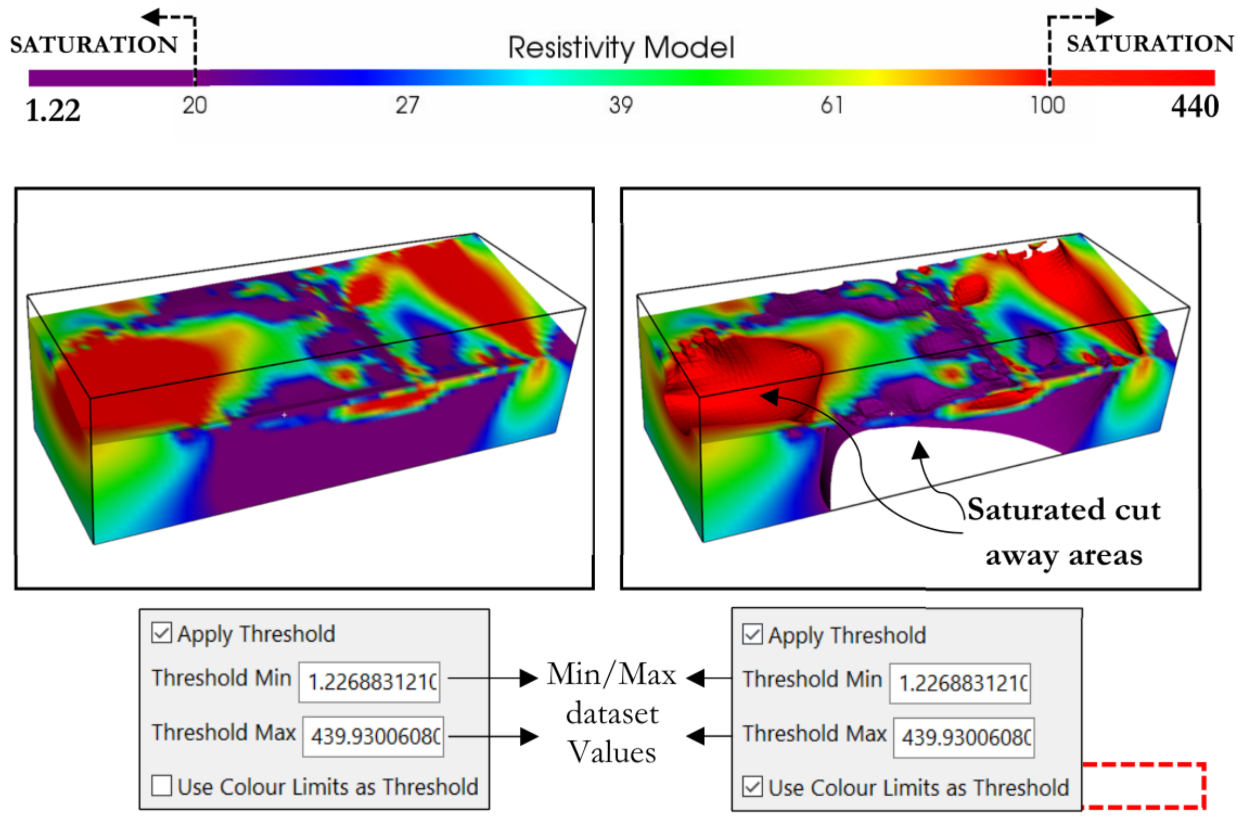


Figure 16.5: Example of full resistivity model with and without 'Use Colour Limits as Threshold' applied

Selecting determinate ranges of values, it is easy to estimate what parts of the investigated area (and in what proportions compared to the entire dataset) have a maximum, medium or minimum resistivity values.

16.6 Smoother

Smooth the volume in case they appear to edgy.

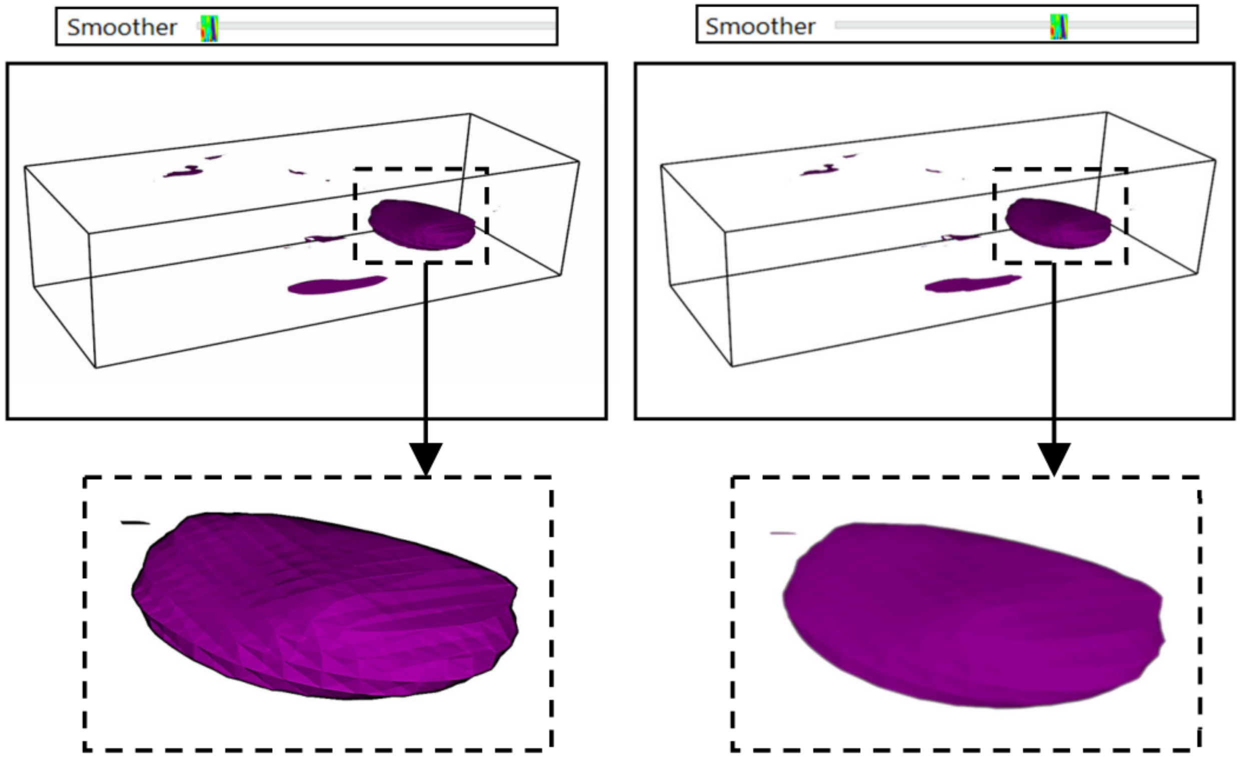


Figure 16.6: Example of smoother applied to a resistivity volume

CHAPTER SEVENTEEN

MODEL ISOSURFACE

This option allows to identify the surface in the model with the same resistivity/conductivity/IP/sensitivity values, the iso-surface. The window has the following items:

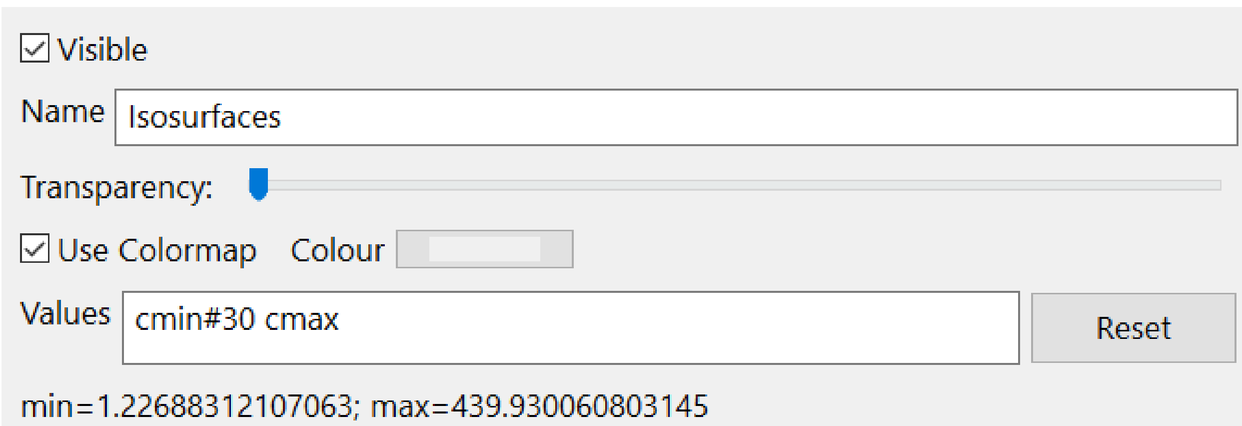


Figure 17.1: Surface panel

17.1 Visible

Sets the visibility of the *Surface*.

17.2 Name

Assign to the *Surface* (default name) the desired name by typing in the text box.

17.3 Transparency

Manages the transparency level of the surface. When the slider is totally at the left the object is completely visible. When it is on the right it is not visible at all and at intermediate positions it is more transparent as the slider goes towards the right (Figure 17.2).

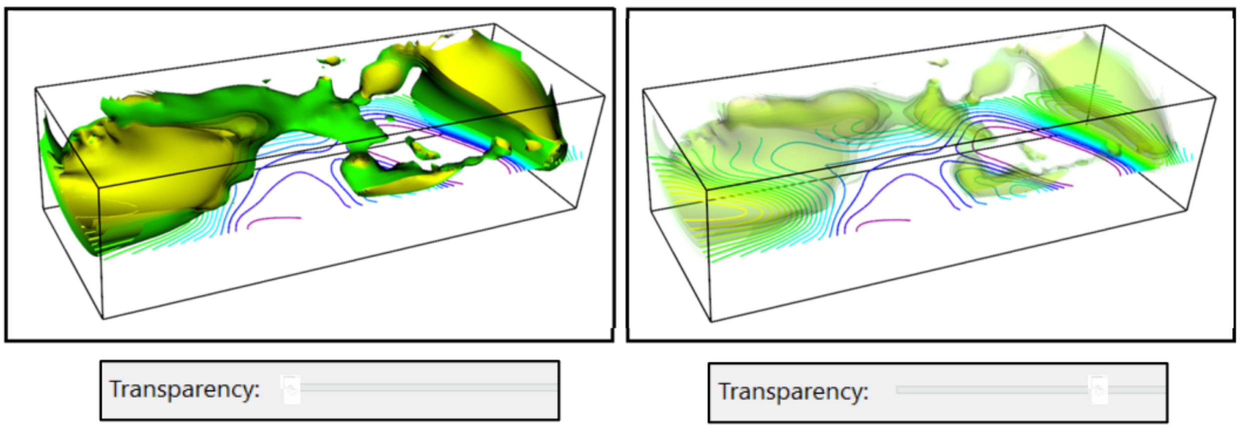


Figure 17.2: Different levels of transparency applied to an isosurface

17.4 Colour

If **Use Colourmap** is checked the iso-surface is displayed with the same colour as the Colour Scale. If unchecked it is possible to choose a colour from the **Colour** button (grey in the following example).

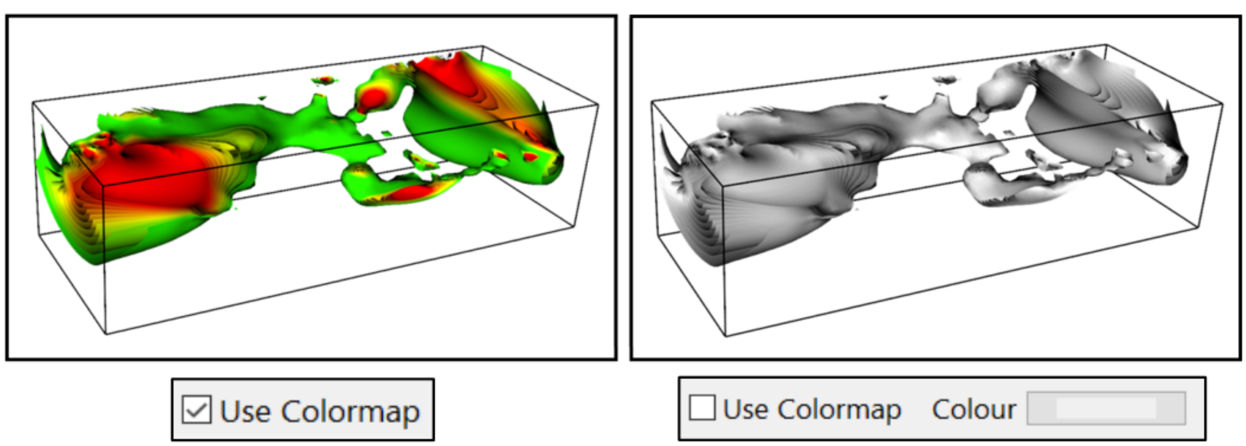


Figure 17.3: Isosurface of one colour (grey) and with the same colormap colours

17.5 Values

Sets the values of the iso-surface to display. It can be a unique value or a range (see section Mathematical Notations for syntax information). Maximum and minimum values of the data set are suggested at the bottom of the window. By default, it is displaying the entire range of dataset values.

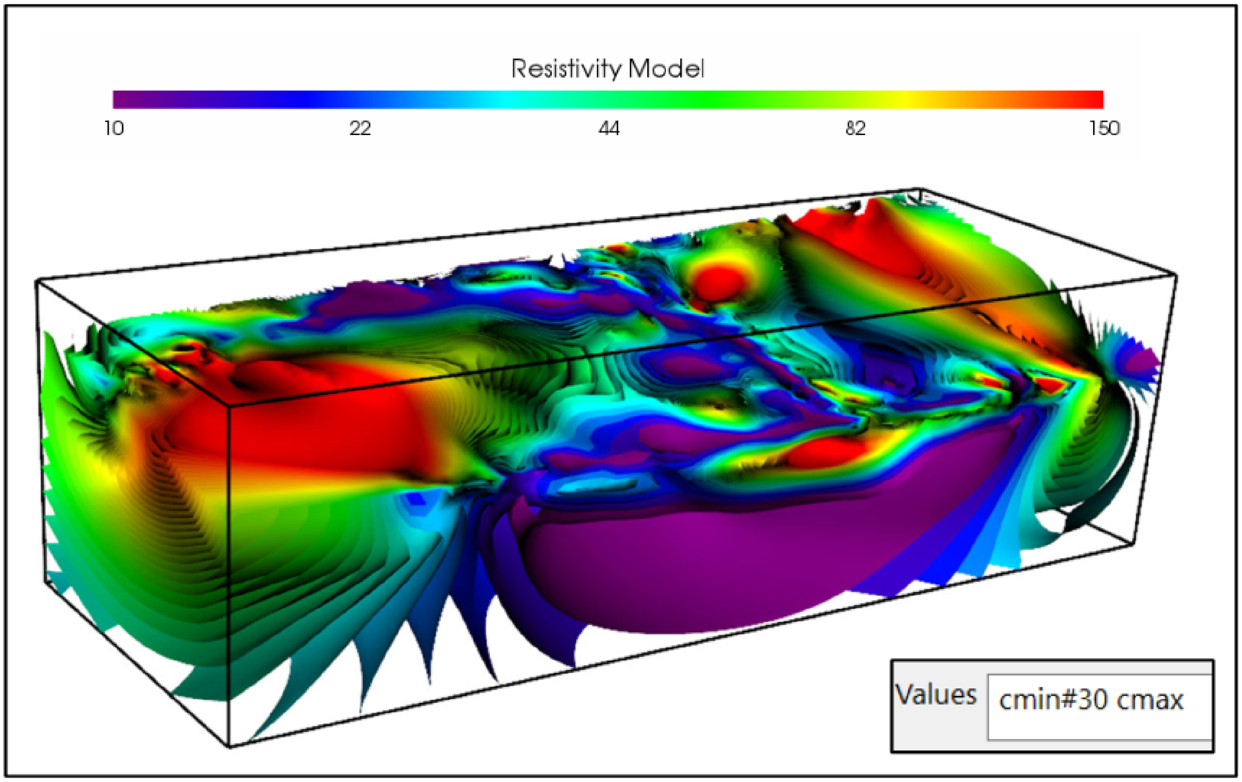


Figure 17.4: Example of setting iso-surfaces values

In Figure 17.5 some specific value ranges are selected : 1-10 (very low resistivity), 20-50 (medium-low resistivity), 60-90 (medium-high resistivity), 120-180 (high resistivity).

In all cases the step values from one surface of the next is 2.

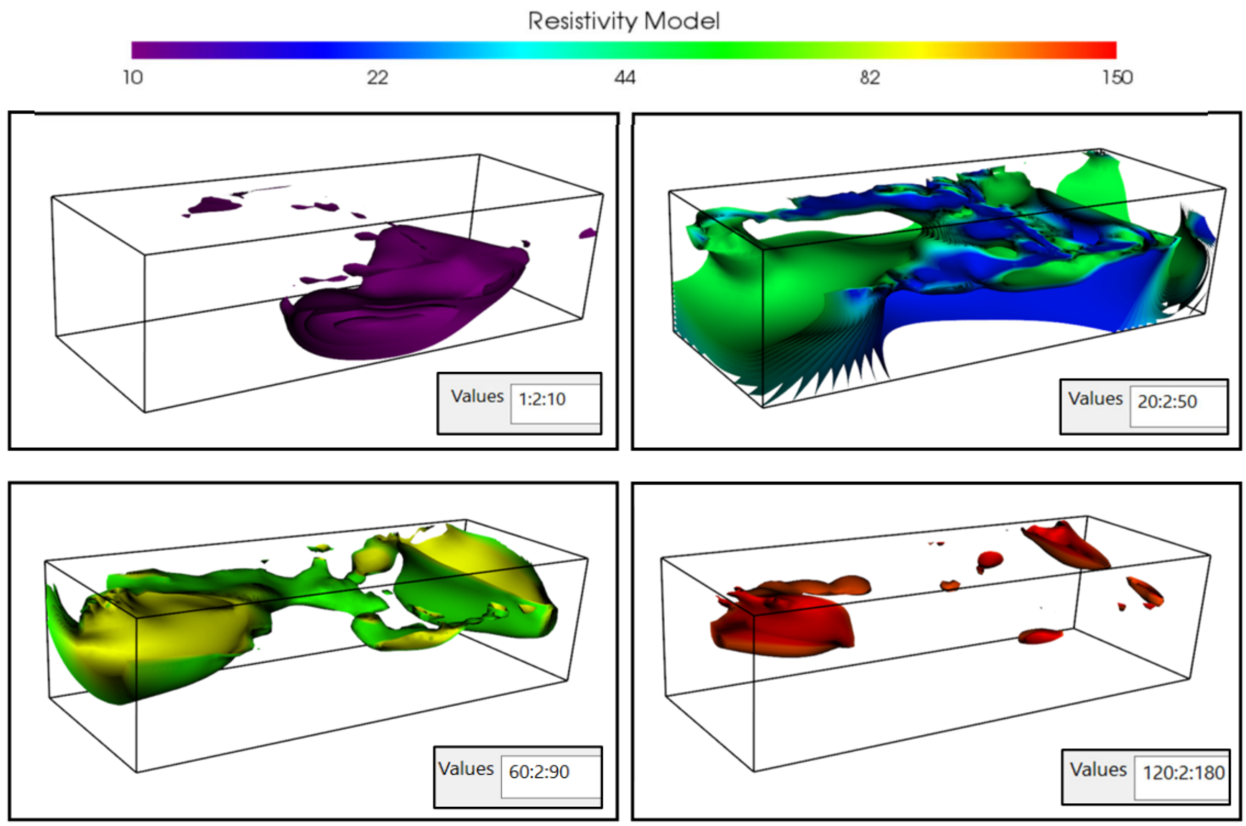


Figure 17.5: Example of different Iso-surfaces setting values

17.6 Multiple

To visualize additional iso-surfaces at the same time add one *Surface* node for each sub-surface range values Figure 17.6.

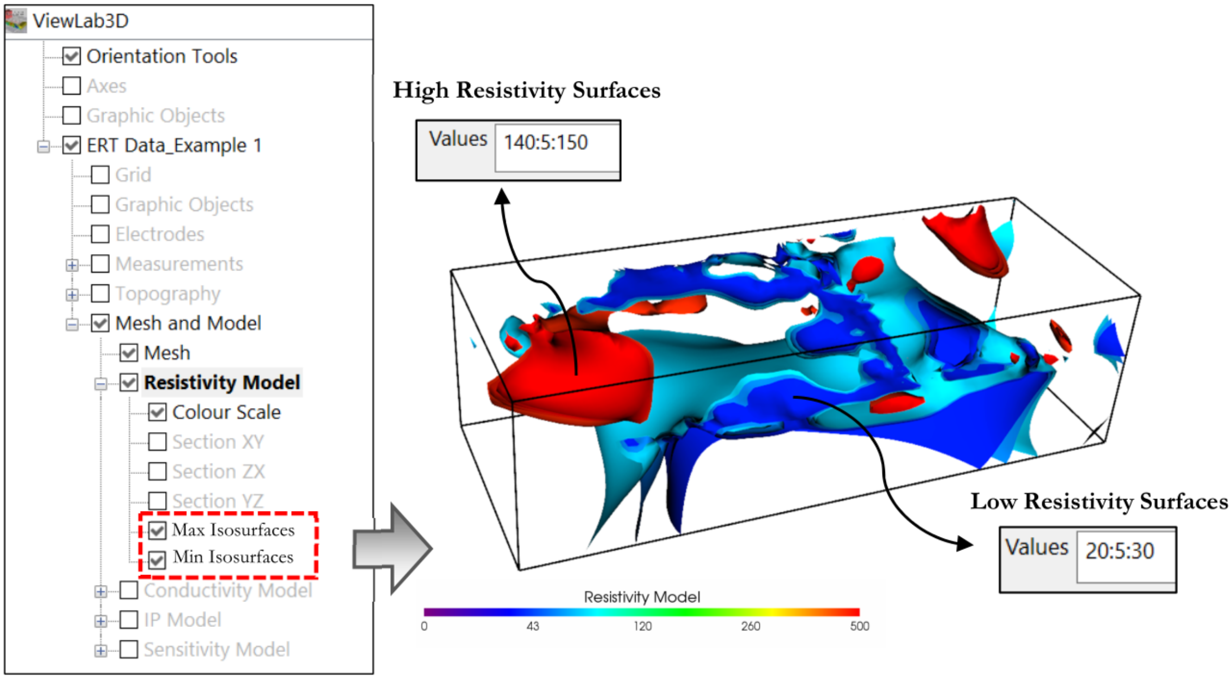


Figure 17.6: Visualization of two Isosurfaces at the same time

CHAPTER

EIGHTEEN

ELECTRODES TOOLS

18.1 Table

By clicking with the right mouse button on the menu option "Electrode", it is possible to manage the electrode information.

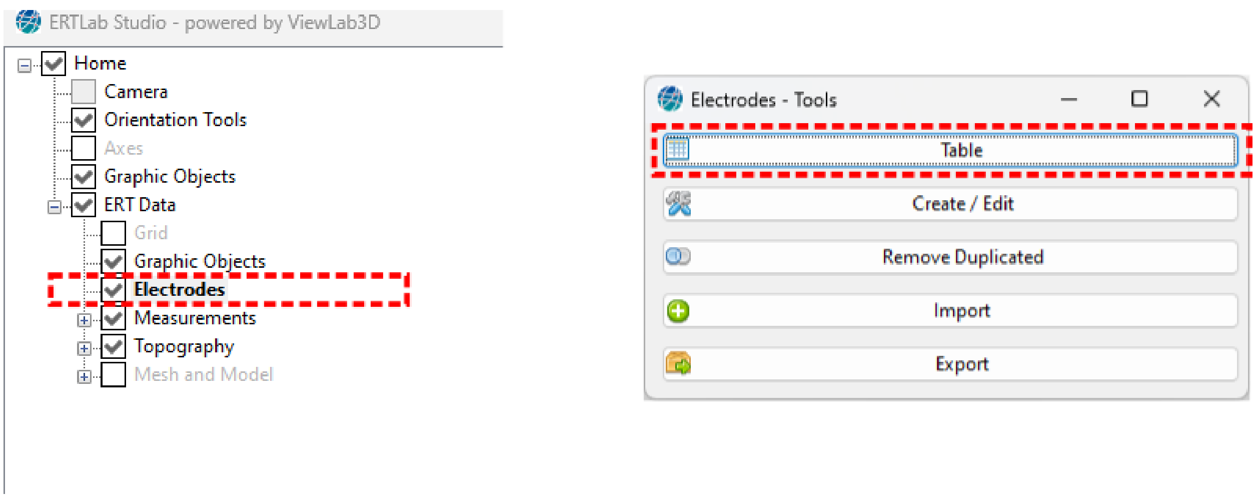


Figure 18.1: Tool Table

It is a table containing the electrode information. By default, the table shows the following columns (Figure 18.2):

Table

	Group	ID	X [m]	Y [m]	Z [m]	Z surf [m]	REM	Skip
1	PD	1	586.01	563.628	314.6	314.6	✗	✗
2	PD	2	584.01	563.628	314.6	314.6	✗	✗
3	PD	3	582.01	563.628	314.6	314.6	✗	✗
4	PD	4	580.01	563.628	314.6	314.6	✗	✗
5	PD	5	580.01	565.328	314.6	314.6	✗	✗
6	PD	6	578.01	565.328	314.6	314.6	✗	✗
7	PD	7	576.01	565.328	314.6	314.6	✗	✗
8	PD	8	574.012	565.223	314.6	314.6	✗	✗
9	PD	9	572.015	565.118	314.6	314.6	✗	✗

Figure 18.2: Electrodes table

- **Group**: name of the cable. It is an editable column.
- **ID**: is a counter that is used to identify each electrode of the group. It is not editable. It's not possible to associate a common "ID" to more than one electrode of the same group, but two or more electrodes of different groups can have the same "ID".
- **X(m), Y(m), Z(m)**: coordinates of the electrodes in the coordinate space. It is possible to change the value of X, Y, and Z by double-clicking the corresponding box.
- **Zsurf (m)**: Z coordinate of the surface (if electrodes are positioned on the surface of the investigated area the Z and the Zsurf have the same value). It is possible to change the value by double-clicking the appropriate box. The two values are different for borehole survey.
- **REM**: flag marking a remote electrode. All electrodes have the red cross symbol except the remote electrode, which is instead identified by a green "V". It is possible to change the flag by double-clicking the corresponding box.

In addition to the default columns, it is possible to add the following columns:

- **TX**: flagged with the green checkmark means that the corresponding electrode works as a transmitter. If it is flagged with a red cross the electrode works just as a receiver. This occurs normally with not-polarizable electrodes as it would be damaged if they would send current.
- **RX**: flagged with the green checkmark means that the relative electrode works as receiver. If it is flagged with a red cross the electrode works just as transmitter. Generally, the electrodes work both as transmitters and as receivers, so both flags are green by default.
- **BOR**: flagged with the green checkmark means that the relative electrode belongs to a borehole survey (and *Zsurf* is different from *Z*). If it is flagged with a red cross the electrode is located on the surface of the investigated area.
- **SKIP**: electrodes flagged with the red checkmark are not used during the sequence generation

process, but they are still in the dataset (they are not deleted).

Note: Note that skipping an electrode does not affect the related measurements. To use electrodes to skip the measurements it is possible to use the command *Skip Measurements using skipped electrodes*.

- **ROLL**: electrodes flagged with the green checkmark are not used during the sequence generation process, because they are supposed to be already used with a previous roll and a different sequence.

If during the acquisition the roll method was adopted, the electrodes common at two consecutive lines can be marked with the green checkmark, otherwise, for normal electrodes they are usually marked with the red cross. In the following example (Figure 18.3), the electrode in blue of the second line (L2) can have "roll = V".

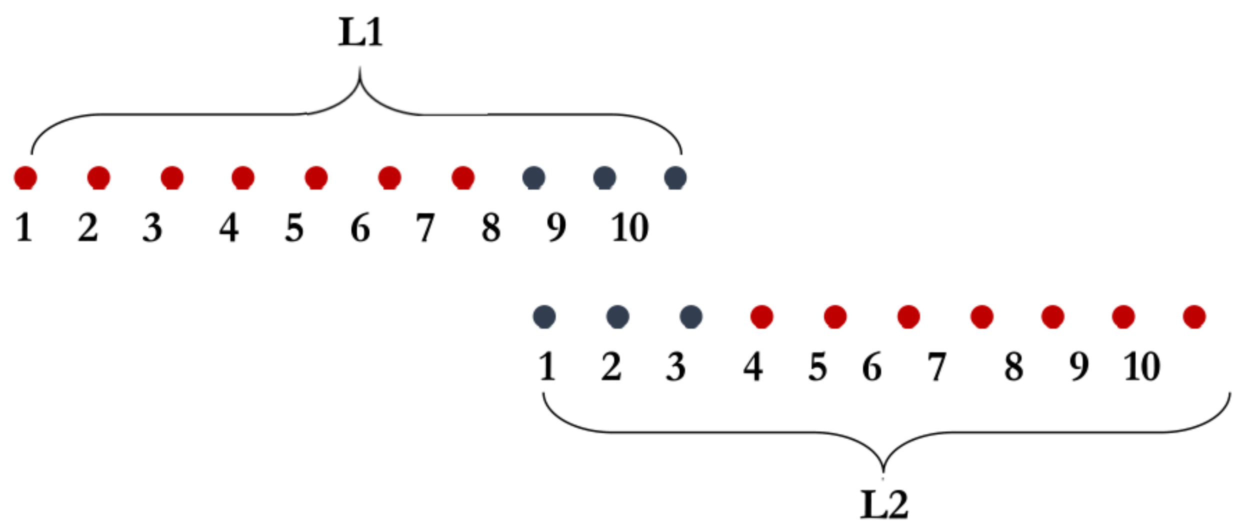


Figure 18.3: Example of roll method

When the line L1 is acquired, the L2 line is positioned moving the electrodes from 1 to 7 and leaving the other electrodes in place. So the electrode 8, 9 and 10 of line 1 becomes electrode 1, 2 and 3 for line 2 (for convenience of representation L1 and L2 are here represented separated but in reality they would be on the same line and the electrodes in blue are not moved). The first six electrodes on line L2 should have the green "V" in the roll column

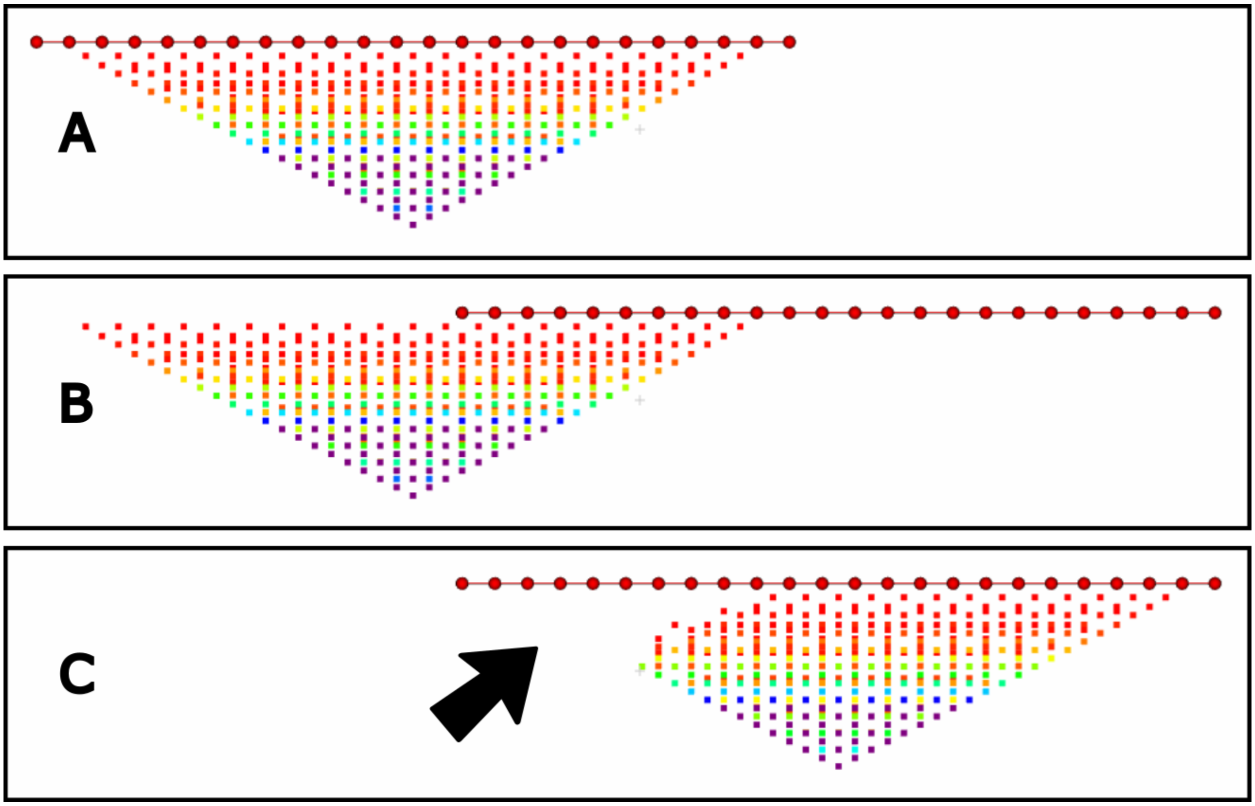


Figure 18.4: Example of roll sequence

In Figure 18.4 it is shown:

- A: a first line of electrodes, with its related measurements.
- B: the second line of electrodes with some common electrodes with the first line. To evaluate the common electrodes the measurements related the first line are also plot.
- C: the second line of electrodes, with its related measurements. The arrow points an empty area obtained during mesh generation setting the common electrodes as *Roll* on the second line.

This is useful to reduce the number of measurements that needs to be collected on the second line (and the following ones), because (if the settings in the Sequence genera panel was set correctly) they should be already collected thanks the acquisition on the first line. The reduction of the number of measurements should also reduce the total time needed in the field for the acquisition of the data, and also to reduce the total energy needed to collect all the measurements.

Left clicking on one of the column headings sorts all data in increasing order of that column. Left clicking a second times on the same column will sort the data in decreasing order.

A popup menu will appear (Figure 18.5) when clicking with the right mouse button in one box. The following options are possible:

18.1.1 Append Row

By clicking the left mouse button in one row (e.g. ID 7 in Figure 18.5) a new row will be added under the selected line (highlighted in yellow in Figure 18.5). By default, it will have the same value of the row from which it was copied from, but it is possible to modify the values.

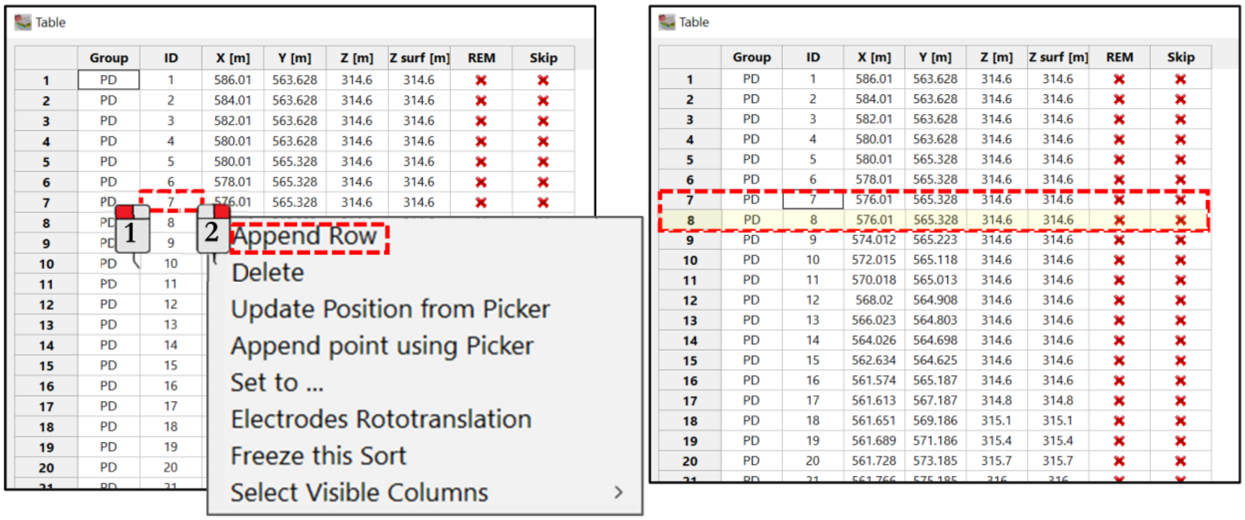


Figure 18.5: "append new raw" in electrode table

18.1.2 Delete

To delete one row, just click with the right mouse button on the row you want to delete and choose *Delete*.

To delete several rows, select the desired rows by holding the left mouse button (the area will be highlighted in blue) then clicking the right mouse button and then "*Delete*". In the following example lines from 5 to 10 are deleted.

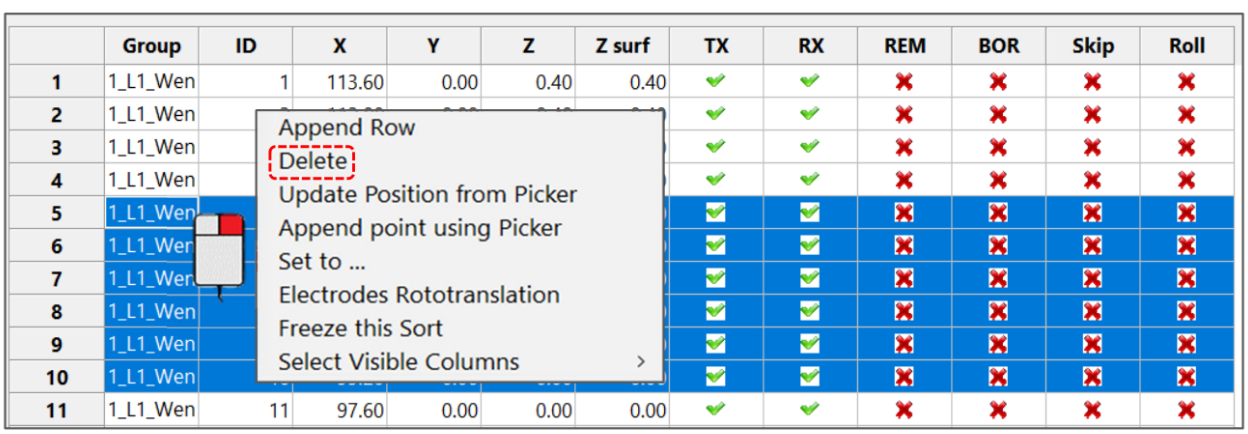


Figure 18.6: Delete some electrodes from the table

18.1.3 Update Position from Picker

Through this tool it is possible to move one electrode from one position to another. If a point in the space has been stored with the *Pick Tool* (see section View Picker settings) his coordinates can be used to replace those already present in one row of the table.

In the following figure, a point is picked in space (pointed out by the arrow in Figure 18.7) at the coordinate:

X = -8m	Y = -6m	Z = 0m

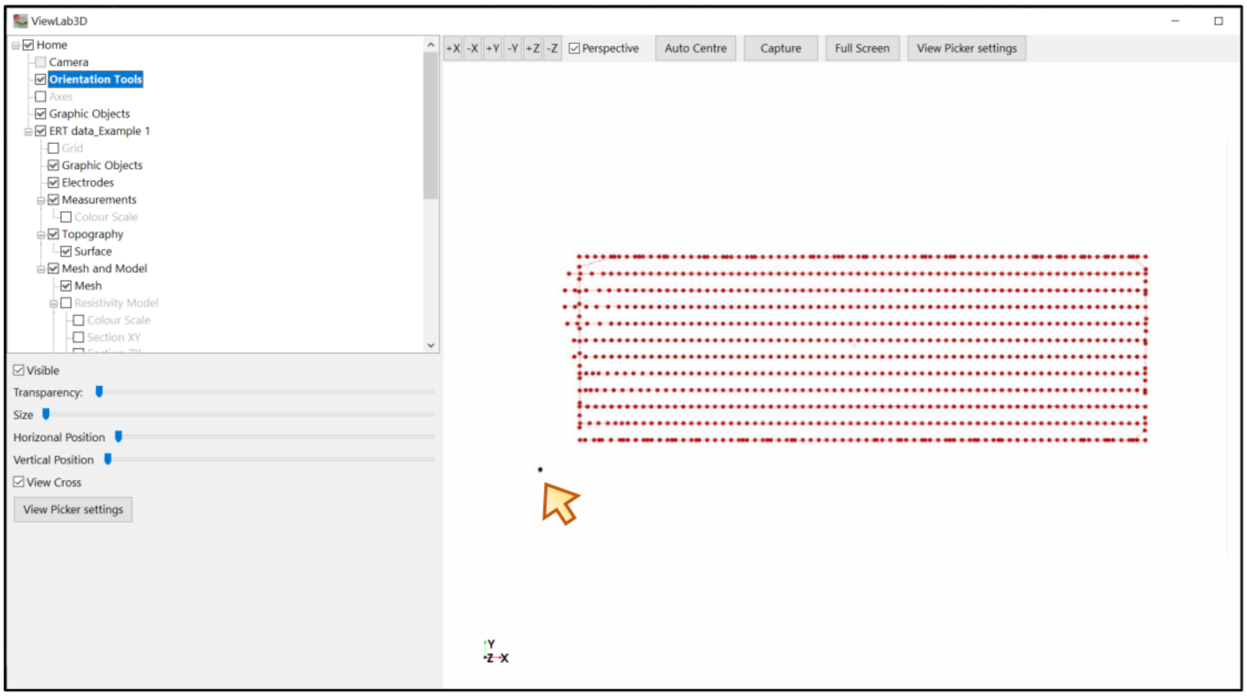


Figure 18.7: One point in the space selected by the picker point

If this point should replace one already present in the electrodes table, click with the right mouse button in the row to replace by choosing "*Update Position from Picker* ". In the following example the picked point (black point in Figure 18.8) replaces the coordinate of the electrode 72 (blue point in Figure 18.8, highlighted by the blue arrow), which was positioned in 0,0,0.

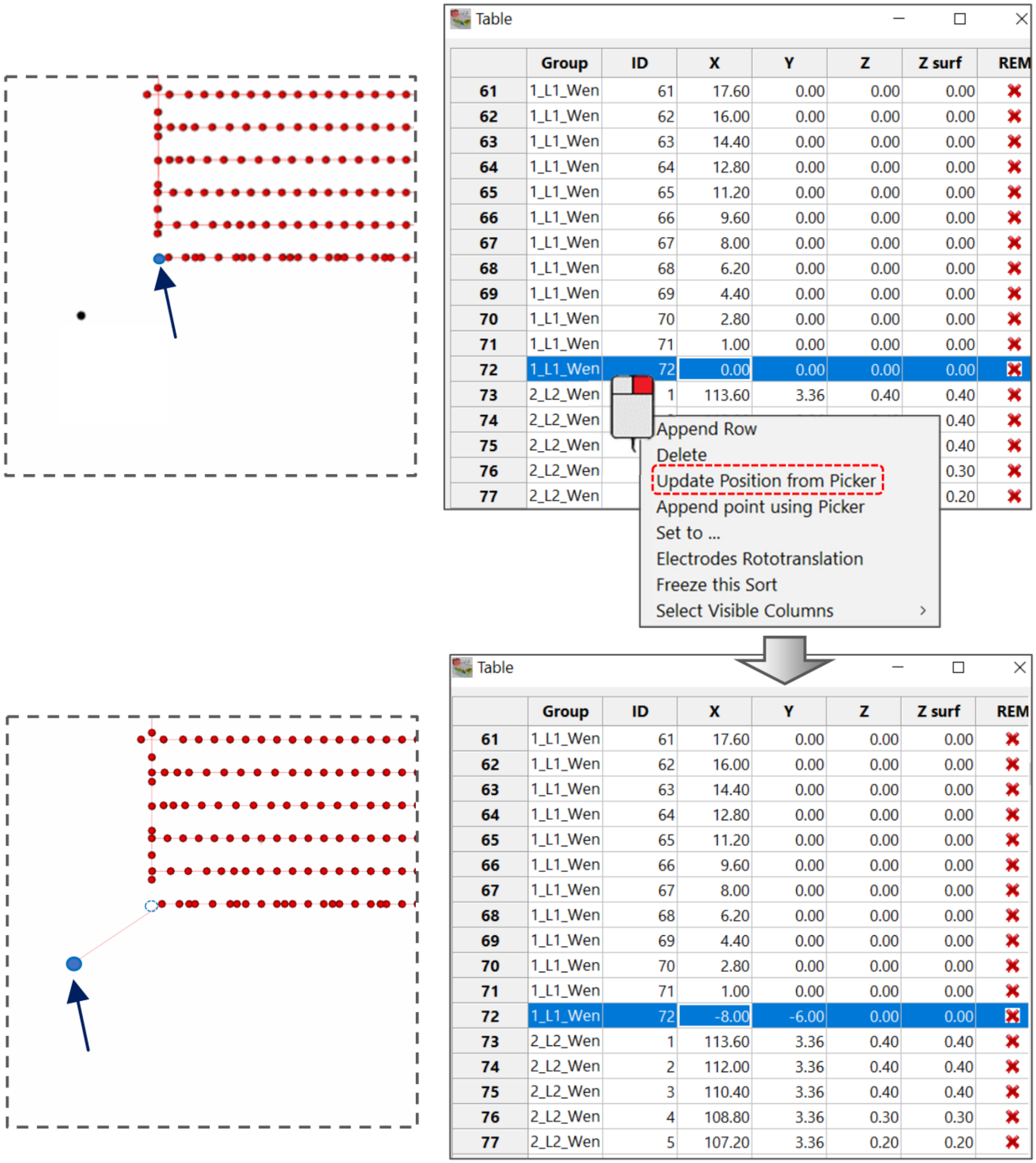


Figure 18.8: Update electrode position using Picker Point coordinates

The electrode 72 was moved to the new position, as is shown in the table and in the plot.

18.1.4 Append point using a picker

Through this tool it is also possible to add one or more electrodes to the existing ones. By clicking with the right mouse button in the row of the table where to add the point and using the coordinates saved with the picker point (as explained in *Update position from Picker).*

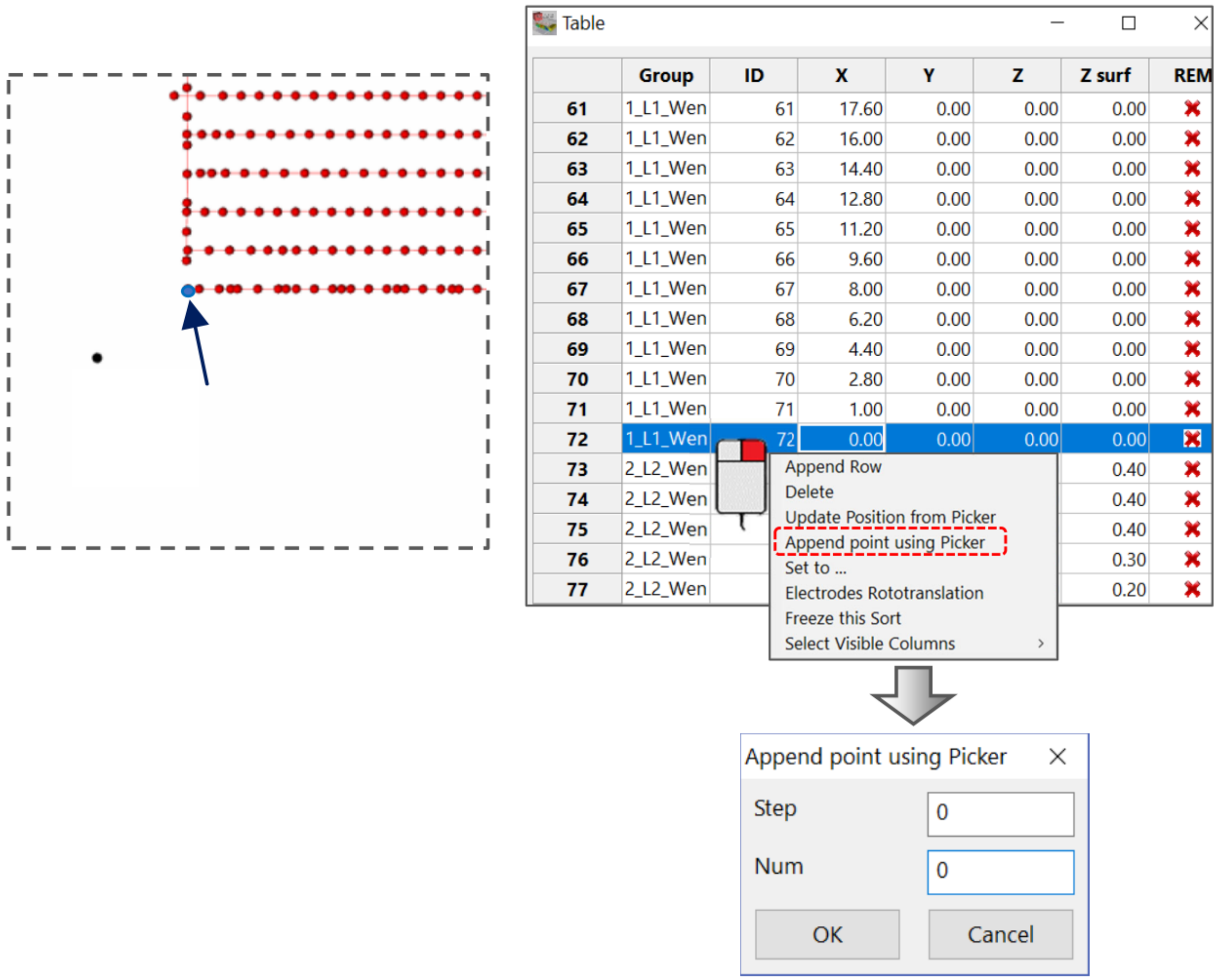


Figure 18.9: A point in the space is selected to add one or more electrodes

In the window that appears (Figure 18.9) the user can choose the *Step* a *Number* values. These values are useful for multiple inserts, where the first point corresponds to the one selected in the table and last to the coordinate of the picker point. "*Step*" determines the space between two points (the number of intermediate electrodes is calculated accordingly), or vice versa. "*Num*" determines the number of electrodes to insert between the first and the last electrode (and *Step* is calculated accordingly). There are different combinations available for the two values:

- **Step=0; Num=0** (default): only one electrode is inserted at the position determined by the picker point (Figure 18.10).

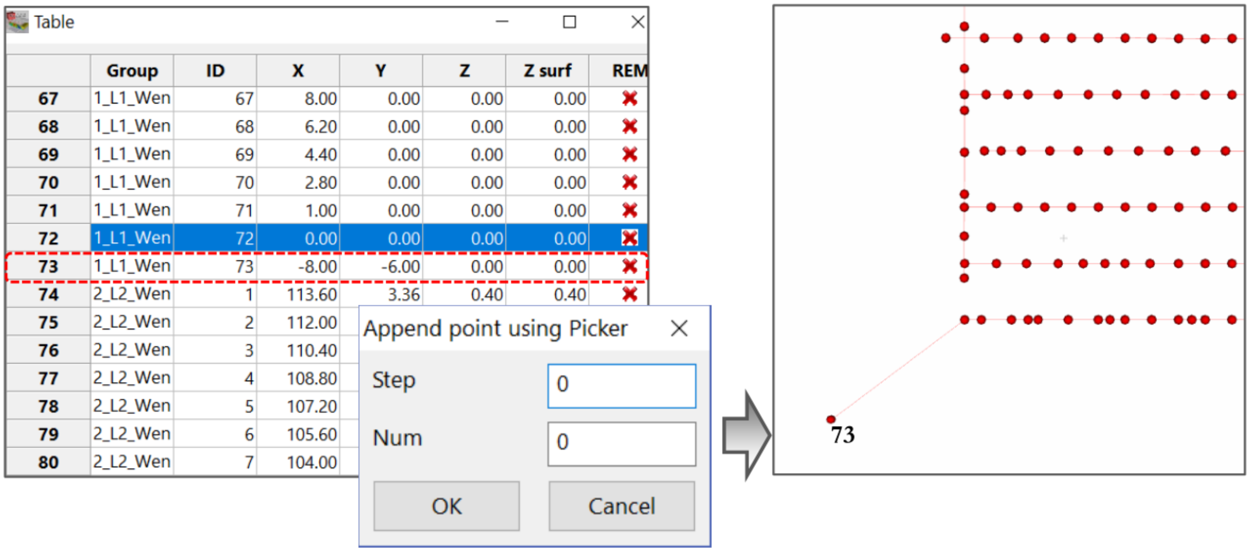


Figure 18.10: One electrode is added to the scene at the picker point coordinates

- **Step= n; Num=0**: some electrodes are inserted in the line, starting from the point of the selected row in the table until the clicked picker point, proceeding with steps equal to the specified values. The electrodes are placed in the scene automatically. As a consequence of the step and the starting-ending position choices the numbers of the added electrodes are automatically found (Figure 18.11).

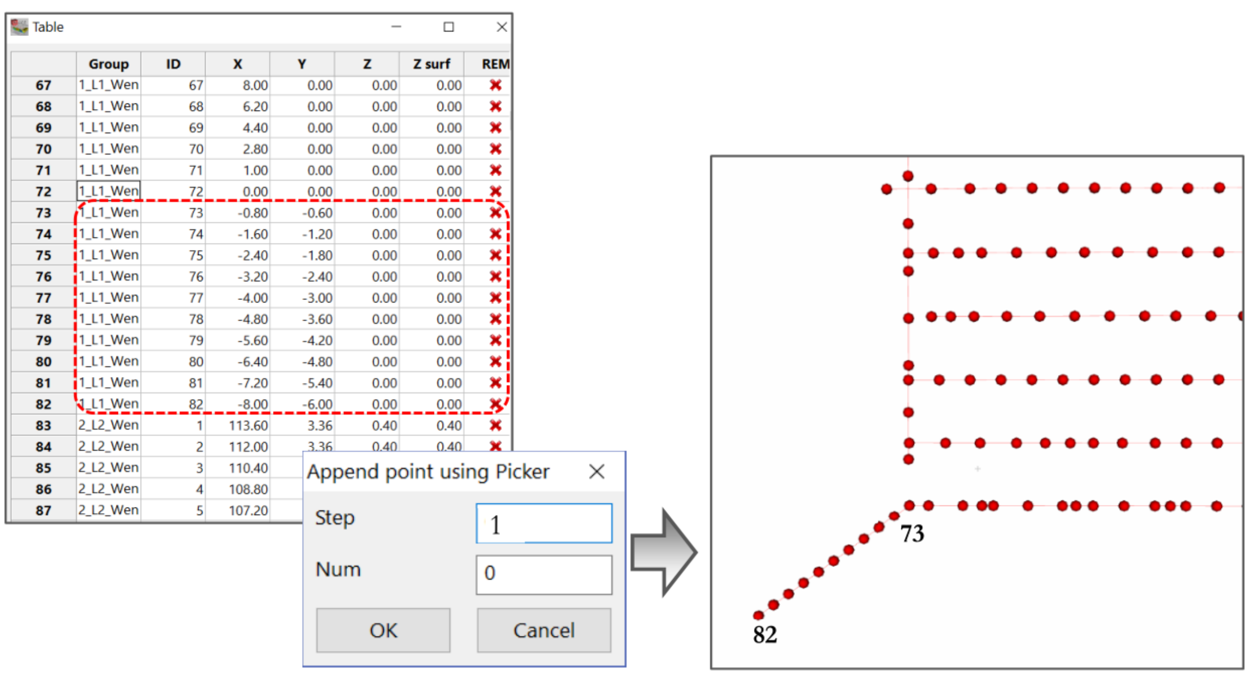
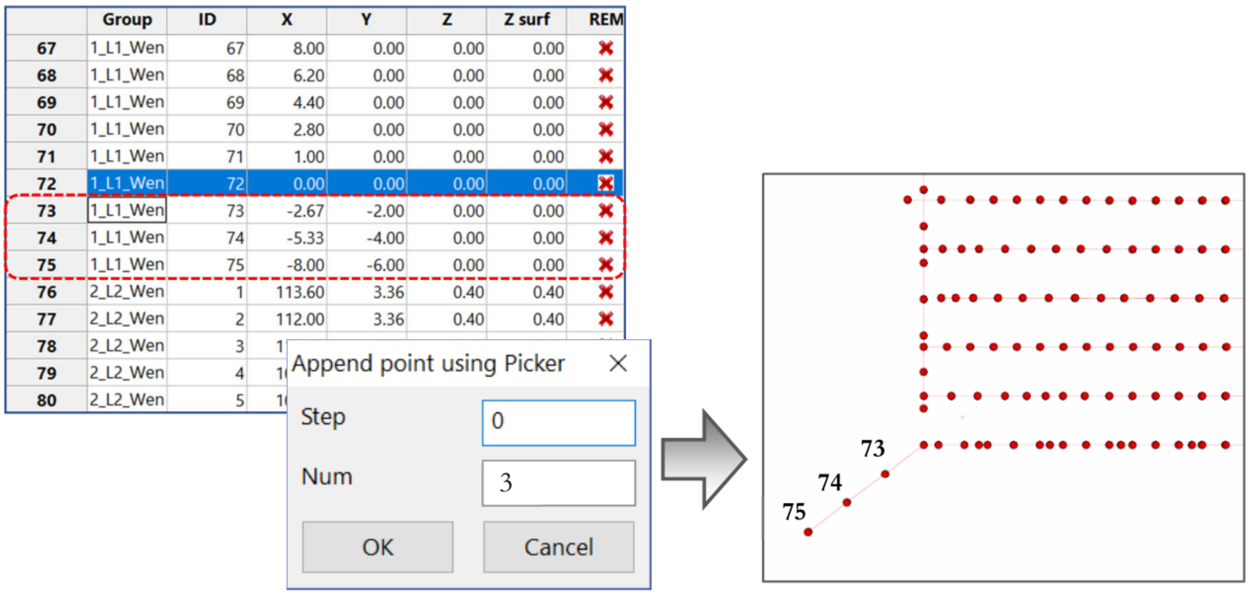


Figure 18.11: More electrodes are added to the scene, choosing the spacing of 1m from each other's

- **Step = 0; Num = n:** a defined number "n" of electrodes is inserted inline, starting from the selected row of the table until the clicked picker point. The sensors are placed in the scene automatically, so the step between them is automatically found.

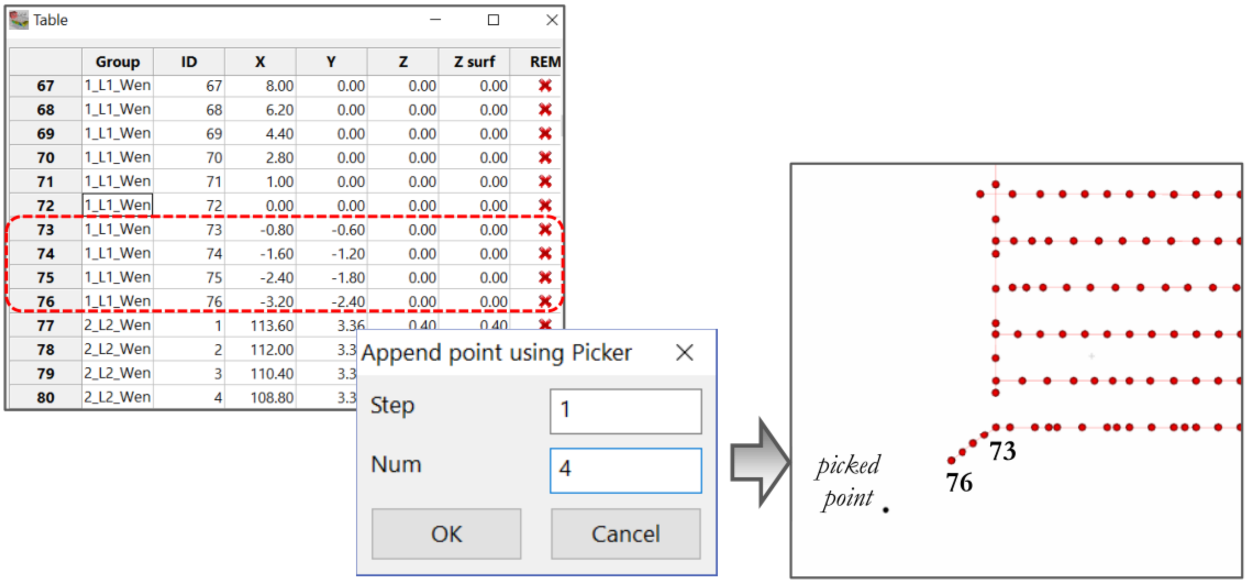
In the following example, 3 electrodes are added from electrode 72 (row highlighted in blue in the table) to 75, which coincide with the coordinate of the picked point.



	Group	ID	X	Y	Z	Z surf	REM
67	1_L1_Wen	67	8.00	0.00	0.00	0.00	✖
68	1_L1_Wen	68	6.20	0.00	0.00	0.00	✖
69	1_L1_Wen	69	4.40	0.00	0.00	0.00	✖
70	1_L1_Wen	70	2.80	0.00	0.00	0.00	✖
71	1_L1_Wen	71	1.00	0.00	0.00	0.00	✖
72	1_L1_Wen	72	0.00	0.00	0.00	0.00	✖
73	1_L1_Wen	73	-2.67	-2.00	0.00	0.00	✖
74	1_L1_Wen	74	-5.33	-4.00	0.00	0.00	✖
75	1_L1_Wen	75	-8.00	-6.00	0.00	0.00	✖
76	2_L2_Wen	1	113.60	3.36	0.40	0.40	✖
77	2_L2_Wen	2	112.00	3.36	0.40	0.40	✖
78	2_L2_Wen	3	[illegible]				
79	2_L2_Wen	4	[illegible]				
80	2_L2_Wen	5	[illegible]				

Figure 18.12: More electrodes are added to the scene, choosing the number of electrodes to add from the position of the selected electrode in the table to the picker point coordinates

- **Step = n; Num = m;** in this case exactly the specified number of electrodes are placed. Spaced by the values specified. It starts from the selected electrode in the table and it proceeds in the direction of the picked point (which can be further by some electrodes, or never been reached).



	Group	ID	X	Y	Z	Z surf	REM
67	1_L1_Wen	67	8.00	0.00	0.00	0.00	✖
68	1_L1_Wen	68	6.20	0.00	0.00	0.00	✖
69	1_L1_Wen	69	4.40	0.00	0.00	0.00	✖
70	1_L1_Wen	70	2.80	0.00	0.00	0.00	✖
71	1_L1_Wen	71	1.00	0.00	0.00	0.00	✖
72	1_L1_Wen	72	0.00	0.00	0.00	0.00	✖
73	1_L1_Wen	73	-0.80	-0.60	0.00	0.00	✖
74	1_L1_Wen	74	-1.60	-1.20	0.00	0.00	✖
75	1_L1_Wen	75	-2.40	-1.80	0.00	0.00	✖
76	1_L1_Wen	76	-3.20	-2.40	0.00	0.00	✖
77	2_L2_Wen	1	113.60	3.36	0.40	0.40	✖
78	2_L2_Wen	2	112.00	3.3			
79	2_L2_Wen	3	110.40	3.3			
80	2_L2_Wen	4	108.80	3.3			

Figure 18.13: More electrodes are added to the scene, choosing the spacing and the number of electrodes. In this case the last electrode (num 76) does not reach the position of the picker point

18.1.5 Set to

Through this command it is possible to manually edit more than a single cell value in the table.

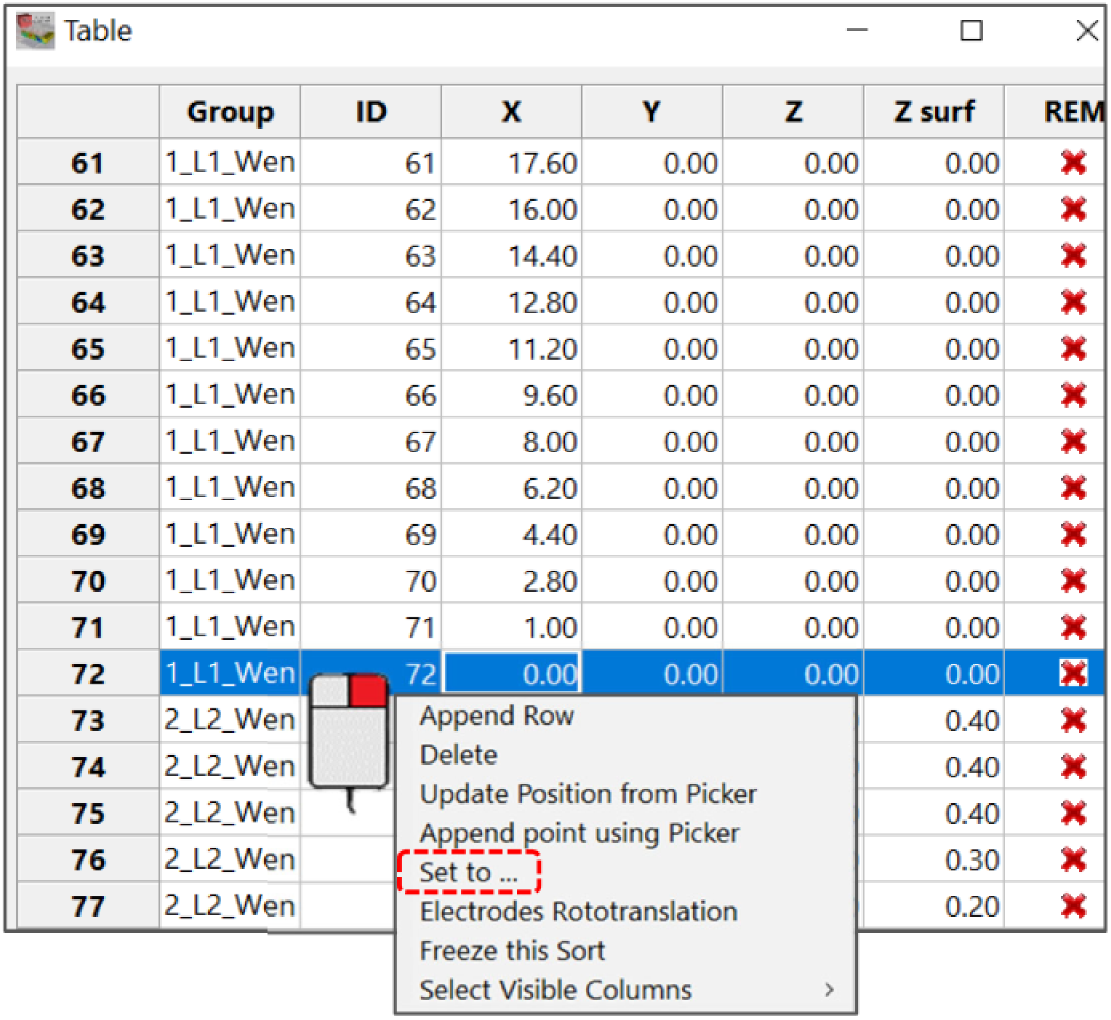


Figure 18.14: The "Set to ..." menu

It is possible to select just one cell and use the "Set to ..." command to change its value, or it is possible to select several cells (in the same column) and use the command "Set to ..." to change them all at once. In this paragraph there is a description of the different results obtained selecting cells from different columns of the table.

- *Group*: selecting one or more cells in this column makes it possible to change the Group Name for the selected electrodes. A window will be shown to ask the user the new name; click on "OK" button to apply. The specified name can be a new group or the name of an already existing group.

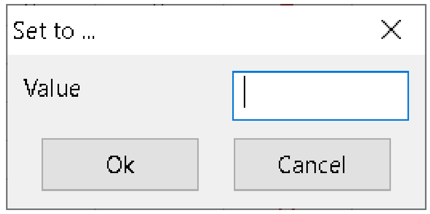


Figure 18.15: Provide the group name

- *ID*: this column is automatically generated, so it is not possible to edit it.
- *X* [m]: selecting one or more cell in this column makes it possible to change the X coordinate of the selected electrodes. A window will be shown to ask to the user the new value(s); to apply, click on "OK" button. If the "*Start Value*" and the "*Stop Value*" are the same, all the cells will be filled with the same value; if they are different, the selected cells will be filled to fit the range specified computing the needed increment automatically.

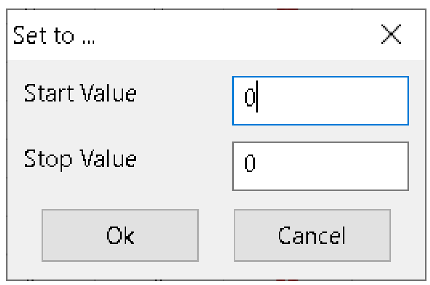
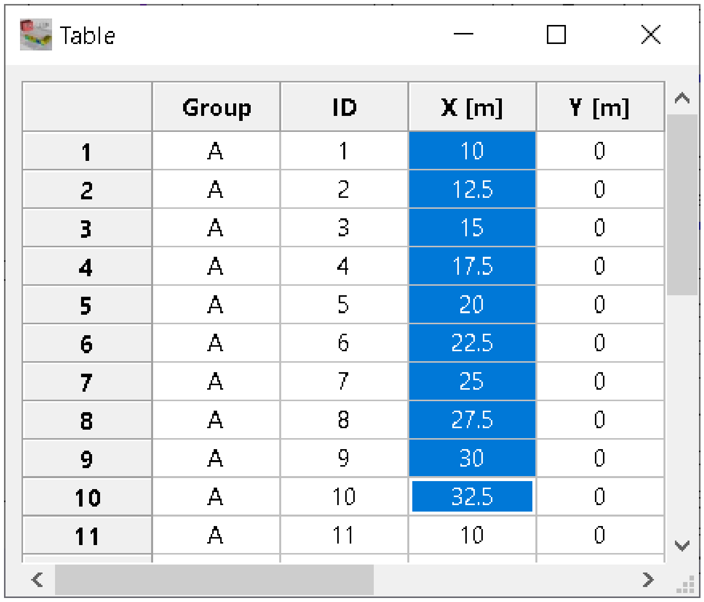


Figure 18.16: Provide the range values

The following image shows an example of how to set a value of 10 as "Start Value" and a value of 32.5 as "Stop value" to a selection of 10 cells. A step of 2.5 and 9 increments between cells are automatically found to satisfy the specified range.



Table

	Group	ID	X [m]	Y [m]
1	A	1	10	0
2	A	2	12.5	0
3	A	3	15	0
4	A	4	17.5	0
5	A	5	20	0
6	A	6	22.5	0
7	A	7	25	0
8	A	8	27.5	0
9	A	9	30	0
10	A	10	32.5	0
11	A	11	10	0

Figure 18.17: Example of setting a linear range of values for cells on “X” column

- *Y* [m]: analogous to X [m] column, see above.
- *Z* [m]: it is almost the same as the “X” column, but with the following difference. The window shown has some more parameter to be checked before continuing.

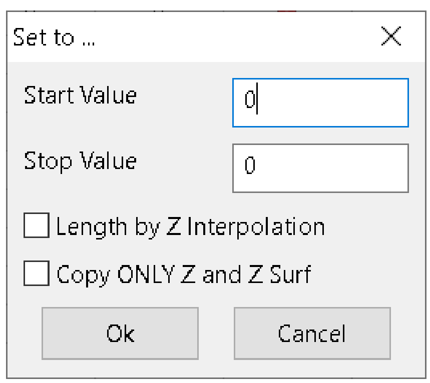


Figure 18.18: Provide the range values

If “*Length by Z interpolation*” is checked the increment automatically found to fill the cells will be not constant but proportional to the electrode distance on the “XY” plane. In the following example 10 electrodes are used, all of them are lying along the “X” axis, the first 5 electrodes are located with a spacing of 1m, then it becomes 4m. In Figure 18.19 A it is shown how the “Z” values are computed with the default settings. Note that the gap between the electrodes along the “Z” axis is constant (always 0.8m). In Figure 18.19 B, the effect of the “Length by Z interpolation” tool is shown. The gap between electrodes along “Z” axis is not constant (it is 0.3m for the first 5 electrodes and 1.2m for the other electrodes which are more distant from each other). The graphic achievement is that in the second case the electrodes can be fitted with a straight line.

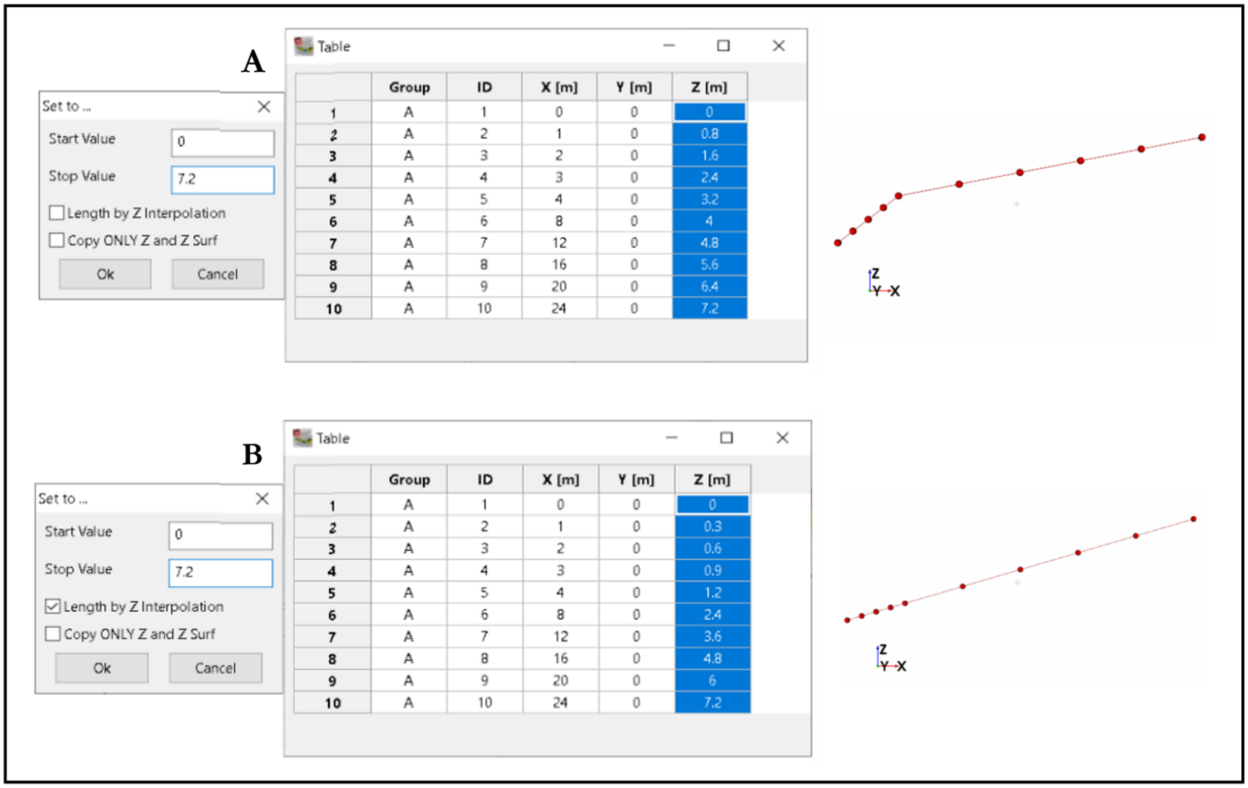


Figure 18.19: Example of using the "Length by Z interpolation" tool

If "*Copy ONLY Z and Z Surf* " is checked, all the other parameters specified in the window will be ignored and the "Z" column will be filled with the values found in the "Z surf " column.

- *Z surf [m]*: same as the "Z" column. The only difference is that when "*Copy ONLY Z and Z Surf* " is checked then the result will be the opposite, so the "Z surf" column will be filled with the values found in the "Z" column.
- *TX, REM, BOR, Skip Roll*: these columns contains binary flags, when the "Set to . . . " command is invoked. The windows shown in Figure 18.20 asks the user the new status that the cells need to have. It is required to check or unchecked the parameter displayed and to press the "OK" button to apply the selection to all the cells highlighted.

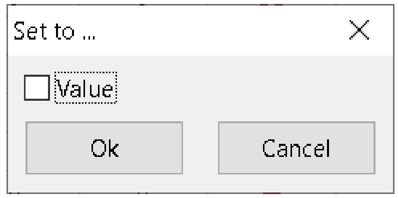


Figure 18.20: Provide the range values

18.1.6 Electrodes Rototranslation

It is possible to rotate or translate the *selected* electrodes. See Rototranslation.

18.1.7 Skip Measurements using skipped electrodes

With this command it is possible to filter the files measurements according to some electrodes that is supposed to give bad data in the field. Note that this menu option is available only when at least one electrode is marked as *Skip*.

For example, if one electrode did not work during acquisition, it is possible to delete the related measurements from the inversion following this process:

- Mark the *Skip* box of the electrodes with the green checkmark, by double-clicking the appropriate box (Figure 18.21, e.g. electrodes 3, 6 and 7);
- Right-click anywhere inside the table;
- Click on *Skip Measurements using skipped electrodes*.

With this function the measurements involving the electrodes marked by the checkmark will be not used for any inversion, but they are still in the dataset (they are not deleted, so it is possible to retrieve them later).

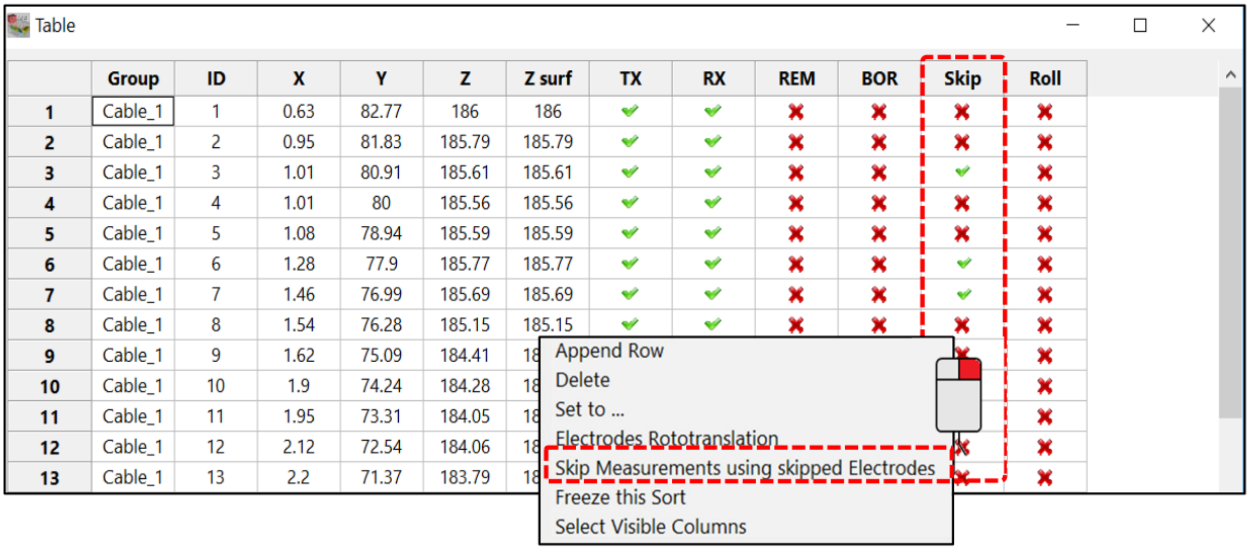


Figure 18.21: Electrodes Table_Skip measurements

18.1.8 Select Visible Columns

With this function it is possible to customize how the table is displayed, so it is possible to show or hide the available columns in the table. It is possible to choose columns one by one or to visualize all of them by selecting "*Show all*", as in the following example (Figure 18.22):

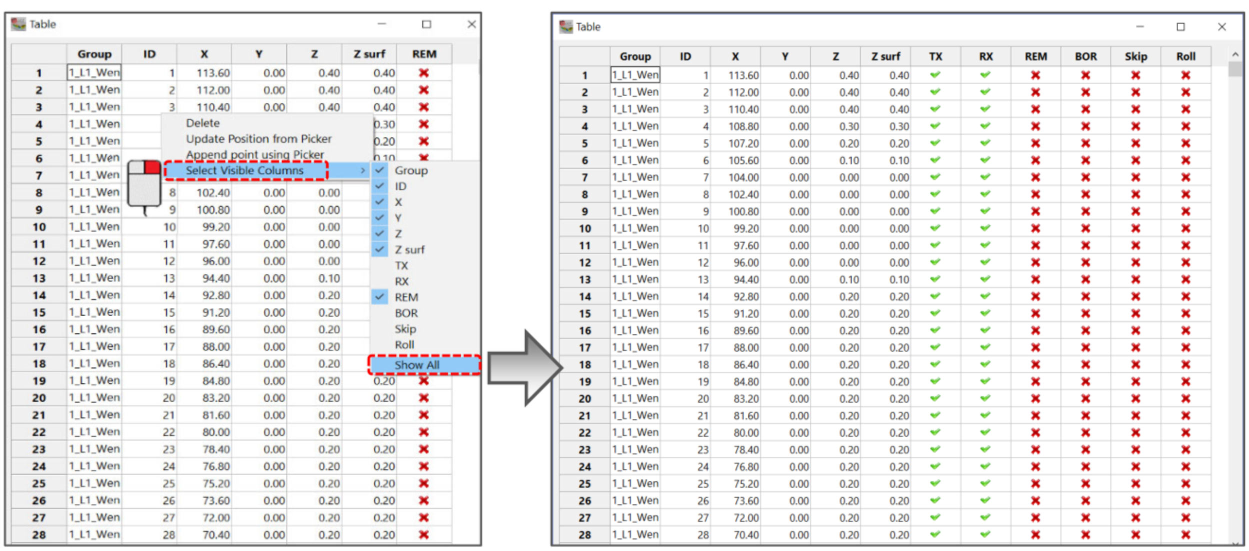


Figure 18.22: "show all" columns in electrodes table

18.2 Create/Edit

Clicking with the right mouse button on the node "*Electrode*", choosing "*Tool*" and *Create/Edit* will open a new window.

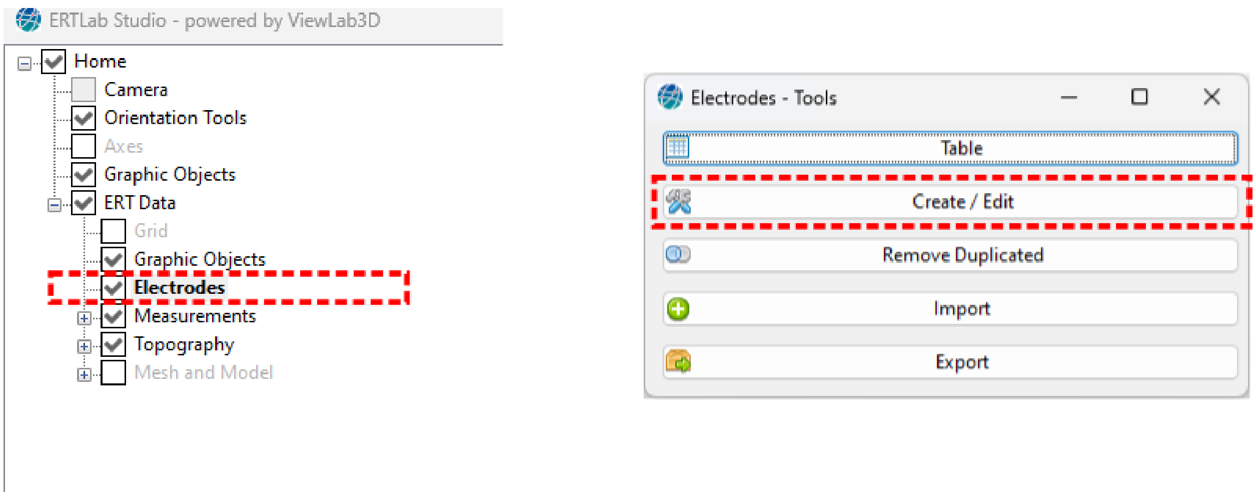


Figure 18.23: Tool Create/Edit

This panel contains a complete subset of tools to manage the electrodes.

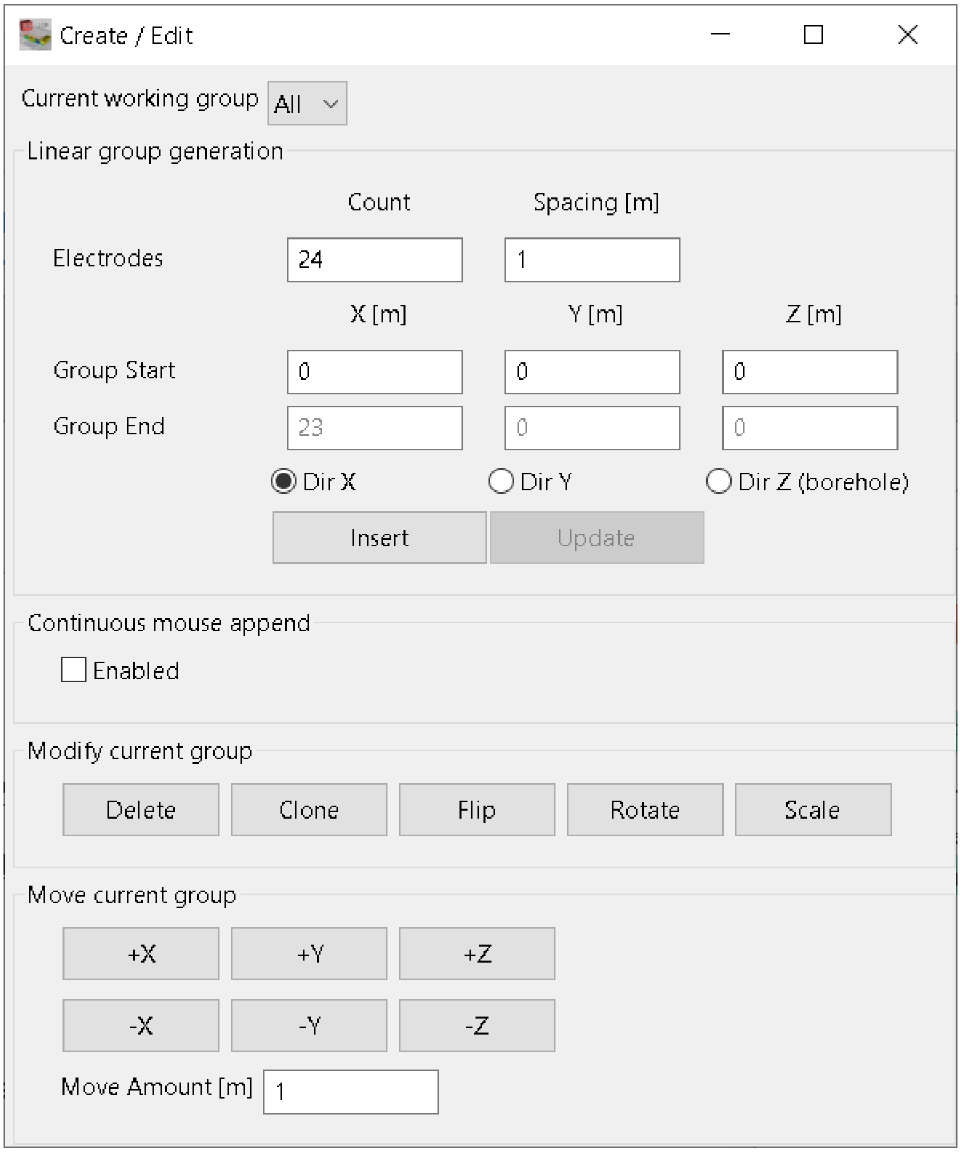


Figure 18.24: Create/Edit tool

18.2.1 Current working group

On the top of the panel there is a selection box that allows to select the Group Name to edit. When the project does not contain any electrode, the previous list contains only the "All" option, else the list contains also all the electrode group names (Group_A/B/C/D in the example below).

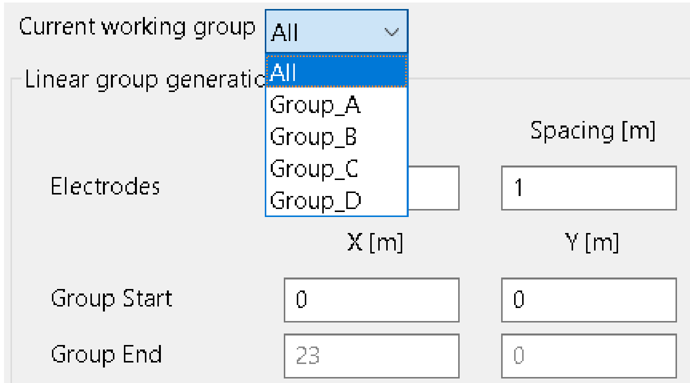


Figure 18.25: Current working group selection

If a group name is selected then any modification that is done in this window will apply to the selected group only. Otherwise, if the "All" option is selected the modification will involve all the groups together (they are managed as a unique big group). In this last case a message is displayed before proceeding.

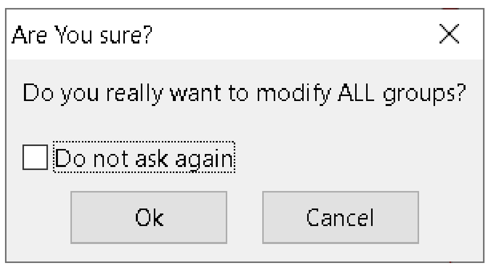


Figure 18.26: Confirm to continue

It is strongly suggested to check the "*Current working group*" every time before applying any modification to avoid an unexpected program behaviour.

18.2.2 Linear Group generation

The following group of tools are related to the creation of a linear array of electrodes. To define the group properties correctly it is necessary to specify some information, e.g. the length of the line, its direction, and its starting/ending point. The tools available in the window lets the user set all these parameters. There are more ways to set the necessary information, but the unset parameters are estimated by the program and displayed for the user benefit.

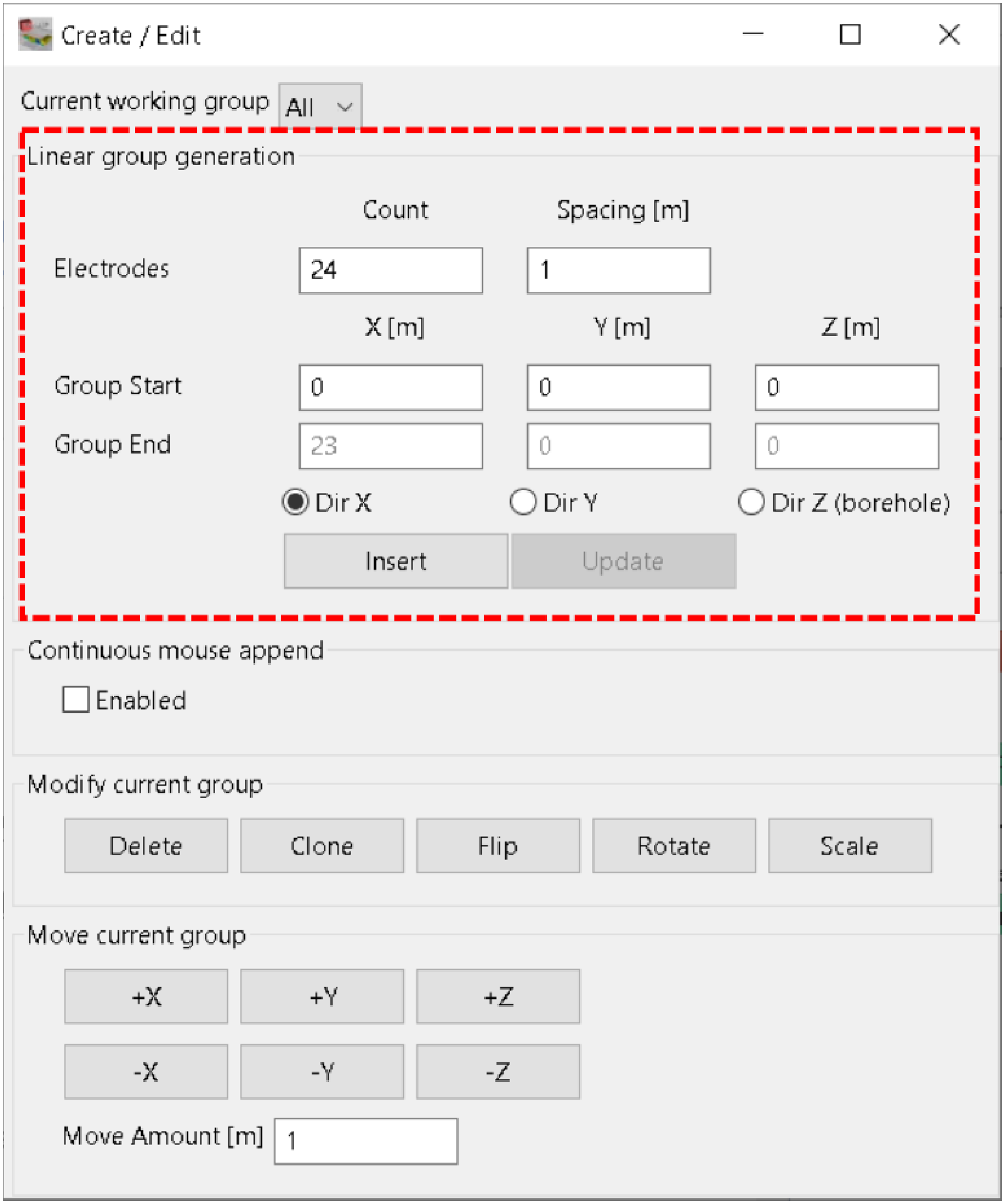


Figure 18.27: Linear group generation tool

- **Electrodes - Count**: specifies the number of electrodes in the linear group. It is related to the instrument used in the field, typically it is a number like 24, 48, . . .
- **Electrodes - Spacing [m]**: specifies the distance between electrodes. Only equally spaced

arrays are accepted.

- **Group Start - X[m]- Y[m] - Z[m]**: specifies the starting point of the linear group in the 3D space. If the user desires to work in local coordinates it is often set to (0,0,0).
- **Group End - X[m] - Y[m] - Z[m]**: specifies the ending point of the linear group in the 3D space. If the user desires to work in local coordinates, Y and Z are set to 0 to have an alignment along the X axis.
- **Dir X - Dir Y - Dir Z**: specifies the direction of the line. It is possible to set only one of them at a time, but in special situations it is allowed to have none of them selected.

Because is it not possible to freely edit all the previous parameters together, the program helps the user to understand the derived values (highlighted with a grey colour) from the ones provided. For example, if "*Count*", "*Spacing*", "*Group Start*" and "*Dir X*" are specified, "*Group End*" is automatically computed.

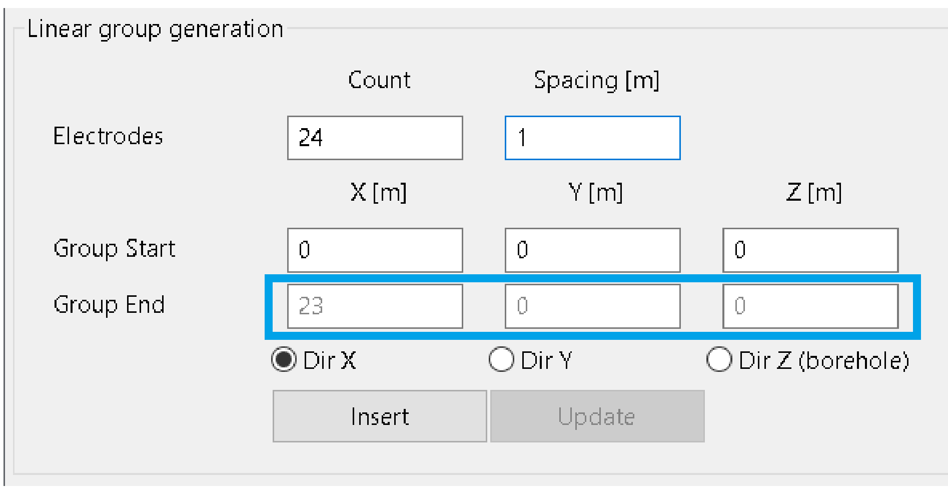


Figure 18.28: An example of linear group definition. Derived values highlighted in the blue rectangle

When the information is provided, click on one of the following buttons:

- **Insert**: a new group will be created. The user will be asked for the name of the new group (it cannot be an already used name). The "*Current working group*" will be automatically set to the new group name to make future editing easier using the other tools available.
- **Update**: the selected group is modified to the linear array specified. It is only possible to use this tool, if a specific Group is selected. If "All" is selected as current working group then the button is not active. If any change of the current working group happens then the values shown in the current sub-panel are updated to fit the selected group features. If the selected group is not linear, the values shown will still be useful to get some important characteristic of the group, e.g. the number of electrodes. When the editing is finished the checkbox must be unchecked to disable the tool and to came back to the default window.

18.2.3 Continuous mouse append

This tool the user can freely add one or more electrodes to the selected group, using a single mouse click.

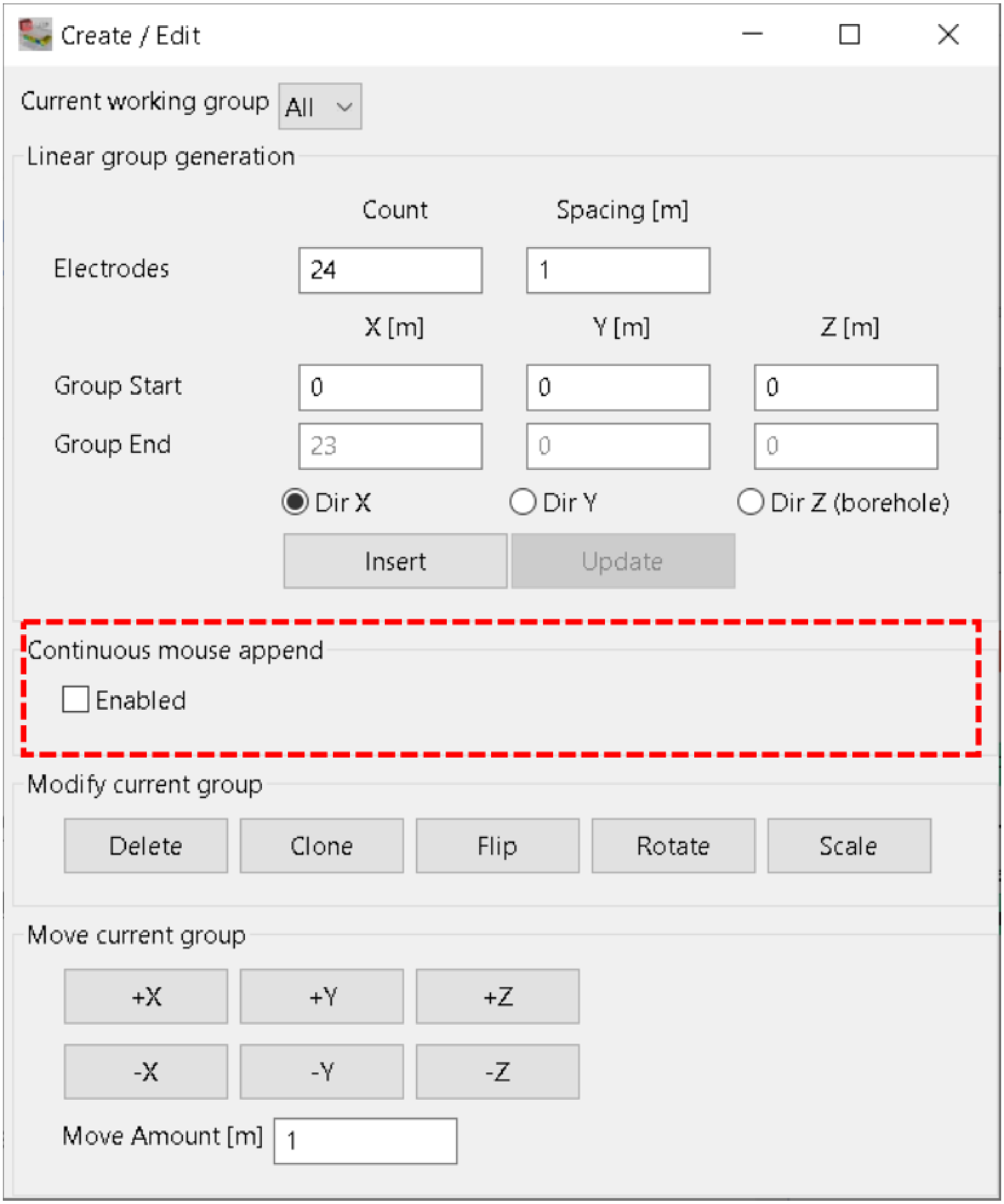


Figure 18.29: Continuous mouse append tools

When "*Enable*" is checked, the tool becomes active and a new window will appear (Figure 18.30). The main panel hides the other tools because they cannot be used at the same time and the window is locked on top of all the other windows to remind the user that this tool is active.

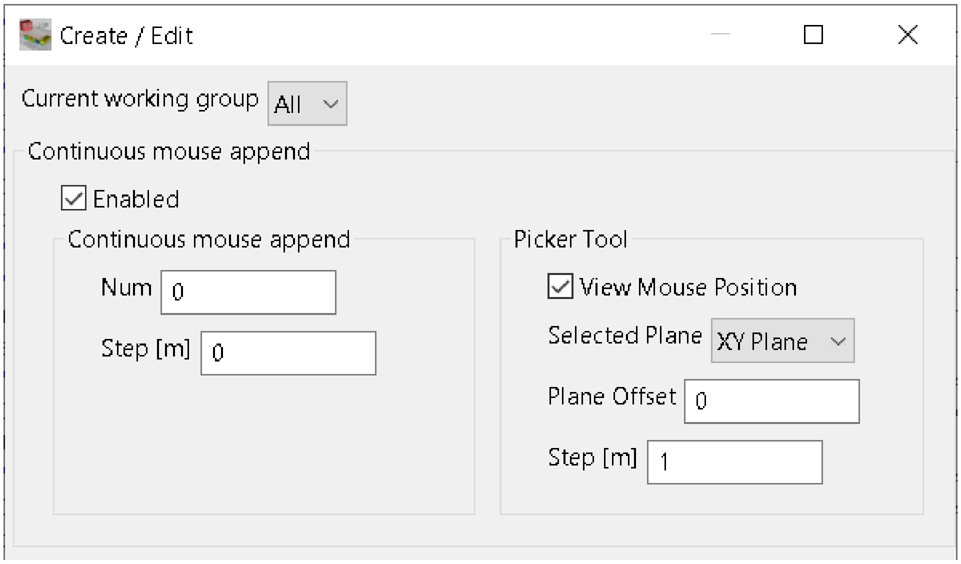


Figure 18.30: The Continuous mouse append tools is active

The new tools of the extended panel was already explained in this document in two places: the first one named "*Continuous mouse append*" is related to the method to add electrodes (section Append point using a picker) and the second one in the "*Picker Tool*", which is related to the visibility of the mouse position (see section View Picker settings). The right approach to use this tool is to select the desired group that needs to be enlarged, then click on the 3D view with the right mouse button to add single electrodes (Figure 18.31). The electrodes are always added at the end of the group.

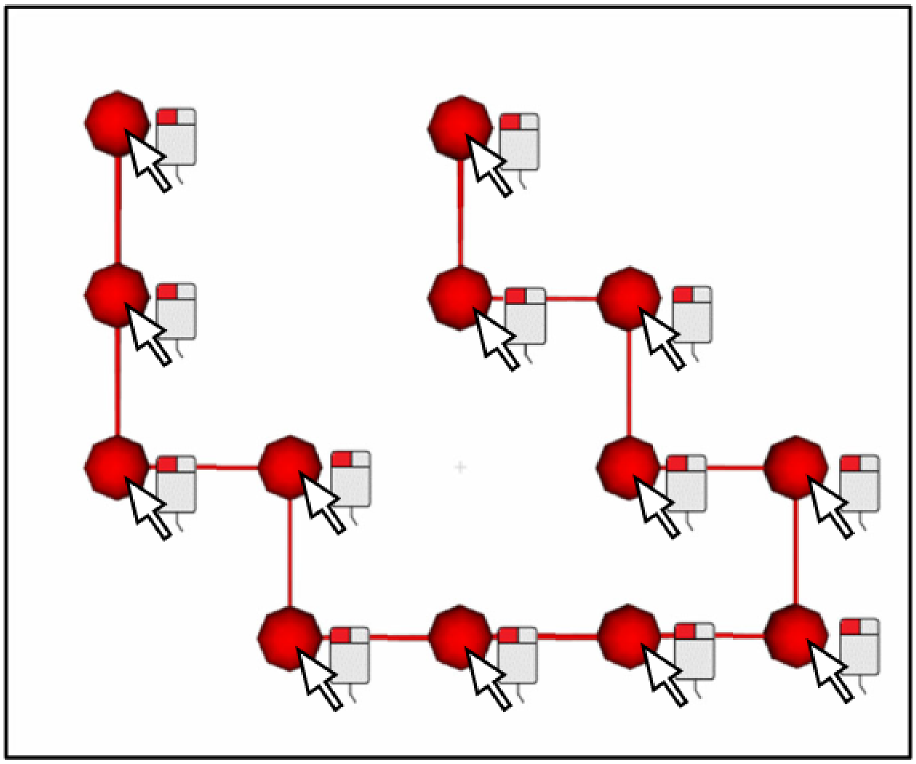


Figure 18.31: An example obtained with single click on every electrode to create in the group

It is possible to add more electrodes with a single operation editing the fields "*Num*" and "*Step*", allowing the user to easily draw a complex layout with only a few mouse clicks (Figure 18.32.

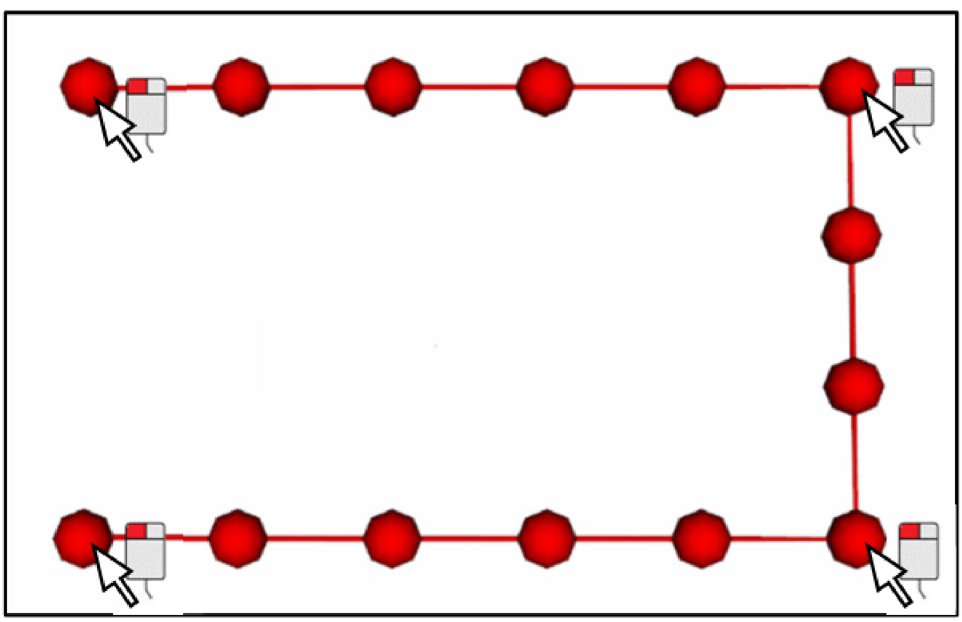


Figure 18.32: An example obtained with few click

If the "All" option is selected as the "*Current working group*", it will create a new group and the user is asked for a new group name.

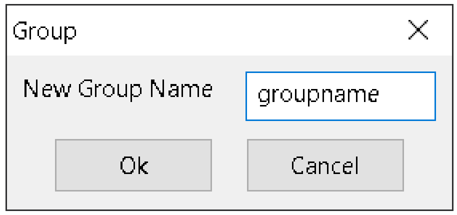


Figure 18.33: Provide a group name

When the editing is completed, the checkbox must be unchecked to disable the tool and to came back to the previous window.

18.2.4 Modify current group

The following group of buttons are related to the modification of existing group(s).

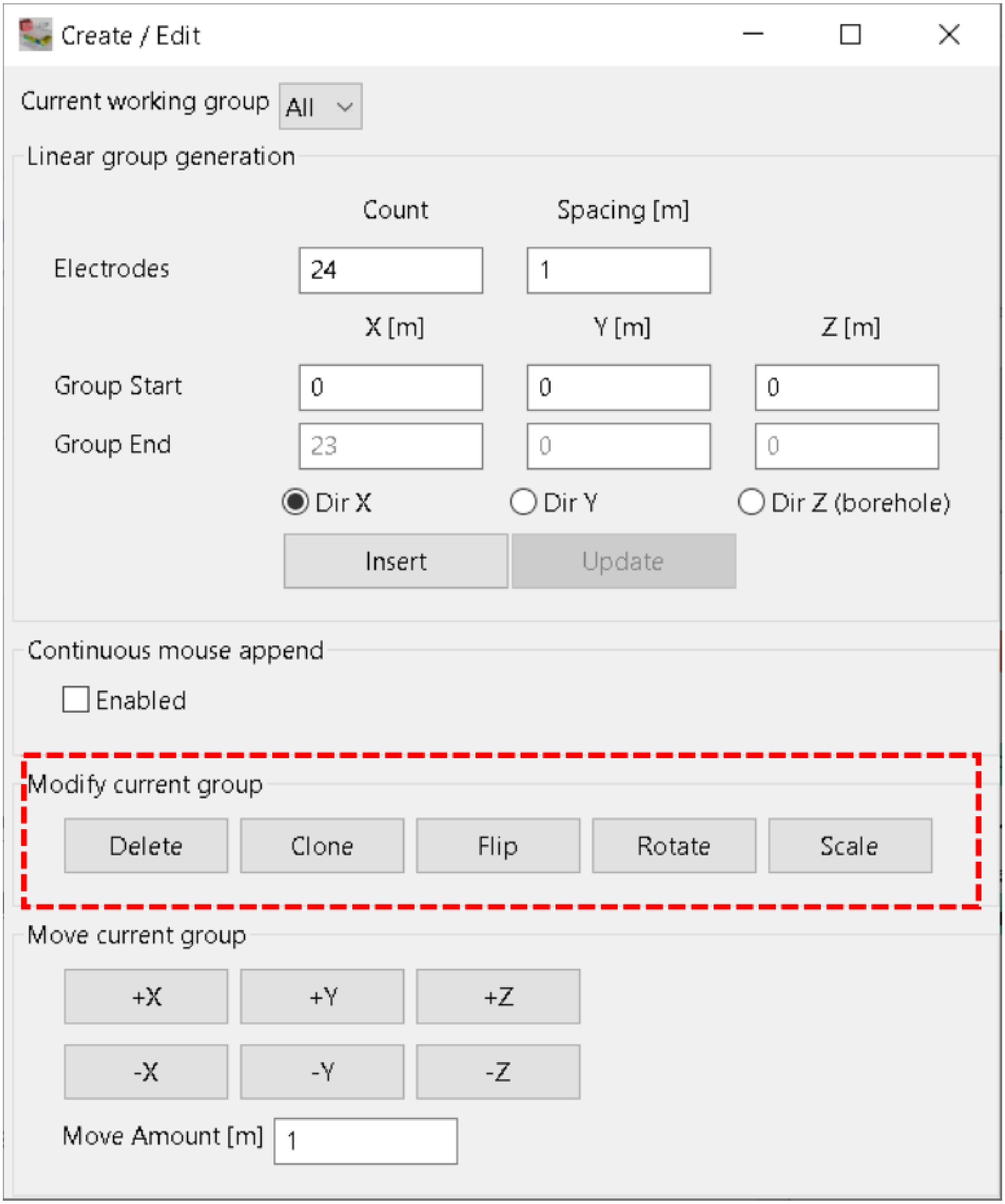


Figure 18.34: Modify current group tools

Delete

Clicking this button the selected group will be permanently removed from the dataset; a confirmation box will be appear before proceeding.

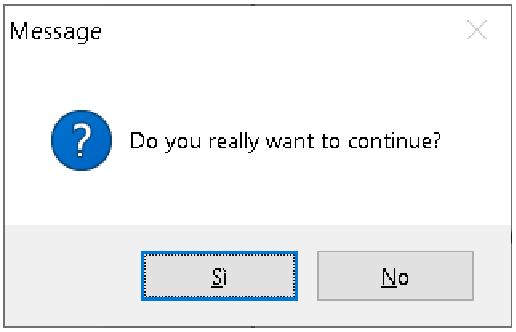


Figure 18.35: Confirm message for "Delete" action

Clone

Clicking this button creates a new group exactly equal to the selected group. The name of the new group must be specified.

The new group will be not easily visible in the 3D scene, because the cloned group source is also displayed. Cloning a group is often not the final target, but a starting point to something more complex: for example, a cloned group can be moved easily to obtain parallel lines.

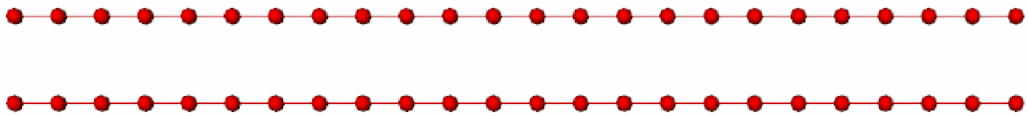


Figure 18.36: An example of a cloned group to obtain parallel lines

Flip

Clicking this button, the electrodes position in the group does not change, but the direction is reversed.

This can be easily verified visualizing the ID of each electrode in the group.

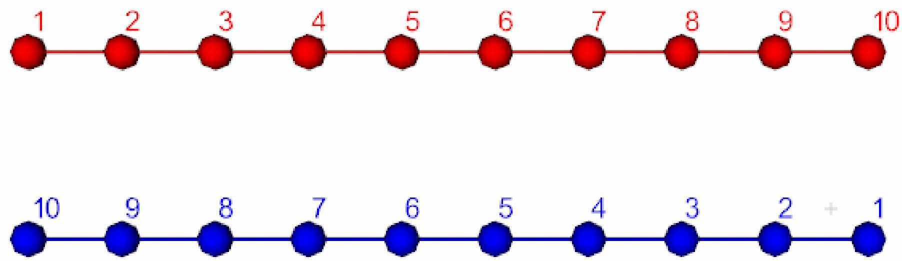


Figure 18.37: The blue group is the flipped version of the red group

This operation is useful to define the electrode geometry and IDs of each cable in the correct way, accordingly to the syntax needed by the instrument that will be used in the field.

Rotate

Clicking this button rotates the selected group. First, it will be asked for an angle of rotation (in degrees) in the range from 0 to 360.

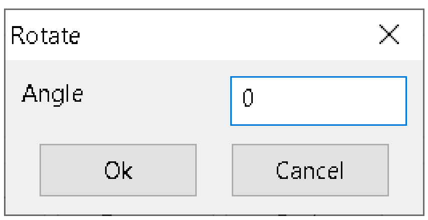


Figure 18.38: It is needed to provide an angle of rotation

Then, the user will be asked for the rotation centre, which will be the barycentre of the group (by default, it is centred in X = 0, Y = 0).

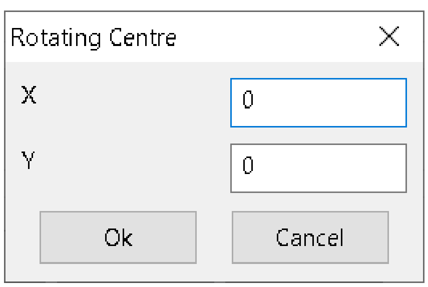


Figure 18.39: It is needed to provide the centre of rotation

Figure 18.40 shows two rotations of 30 degrees; the red group is rotated using his barycentre to obtain the green group, or is rotated centring on its first electrode to obtain the blue group.

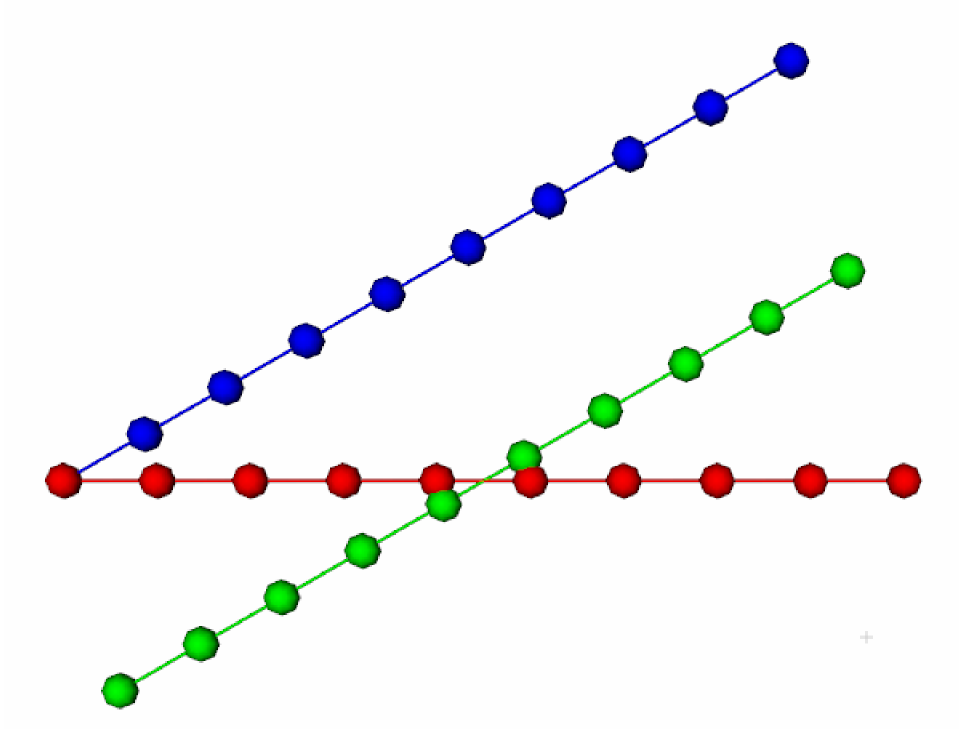


Figure 18.40: An example of group rotation

Scale

Clicking this button changes the size of the selected group. First, the user will be asked for the size ratio change, the default value is 1. A scale factor greater than 1 means that the group will be enlarged, a scale factor smaller than 1 (but greater than 0) means that the group will be made smaller.

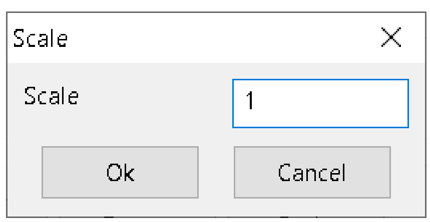


Figure 18.41: It is needed to provide the scale factor

Then, the user will be asked for the centre point for the scaling operation. The message displays the default (X,Y) values, which are the barycentre of the group.

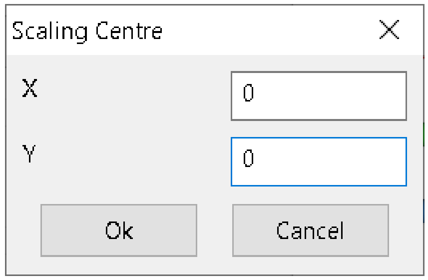


Figure 18.42: It is needed to provide the centre for scaling

Figure 18.43 shows two scaling factors applied: in the first example, the red group is scaled of a factor 2, using his barycentre, to obtain the green group. In the second example, the red group is scaled of a factor 0.5, using as centre point its first electrode, to obtain the blue group.

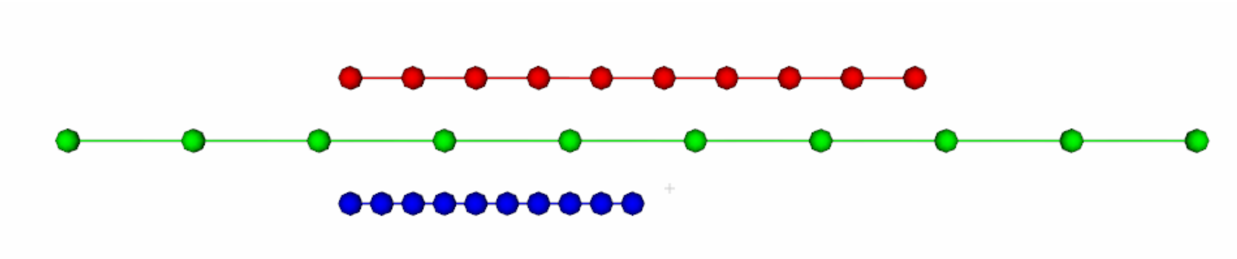


Figure 18.43: An example of the use of the “Scale” tool

18.2.5 Move current group

The following group of buttons can be used to translate group(s) of electrodes along the main axes.

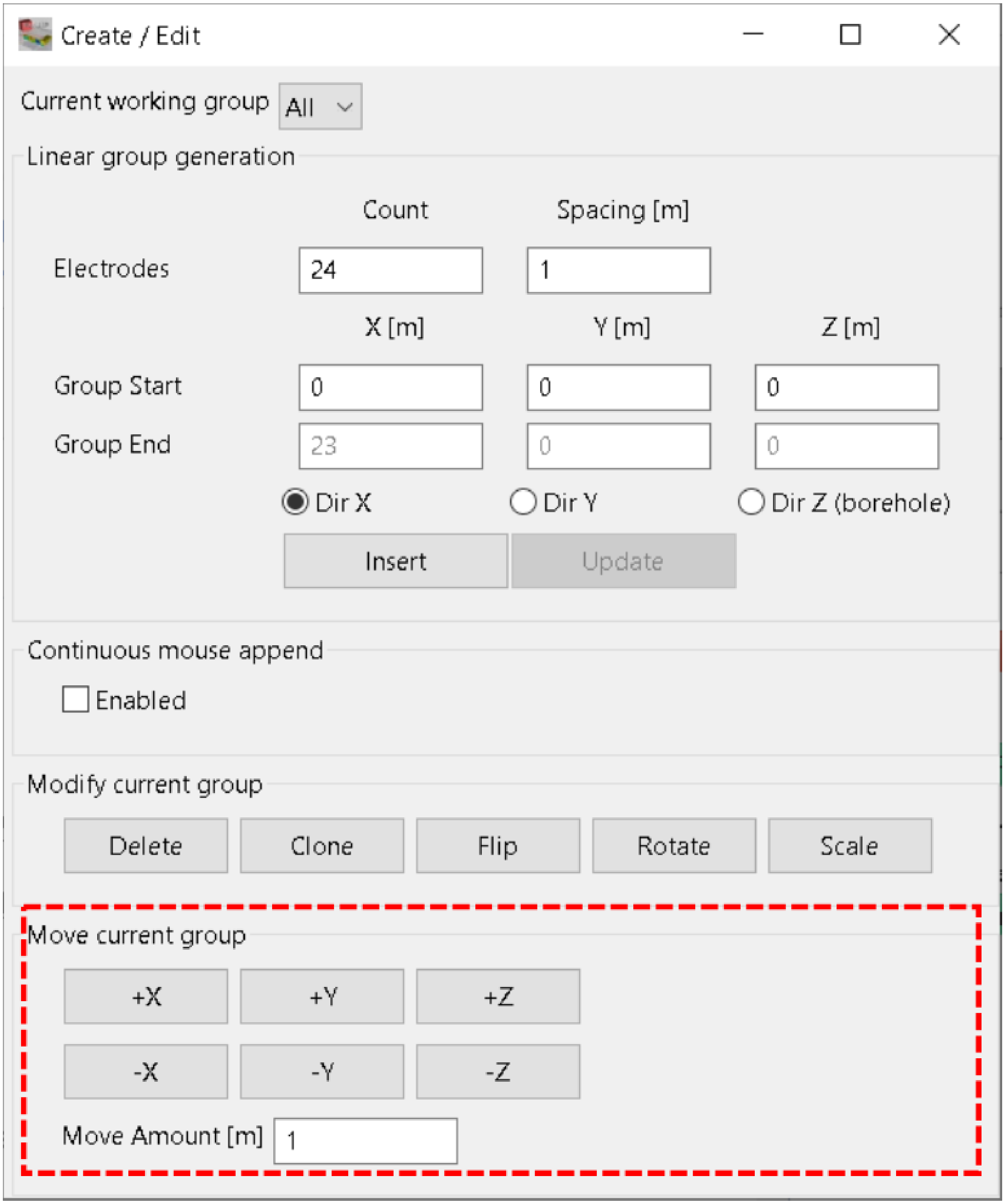


Figure 18.44: Move current group tools

The names of the buttons refer to the axis along which direction to make the translation (the following images helps to understand the type of movement applied by each button).

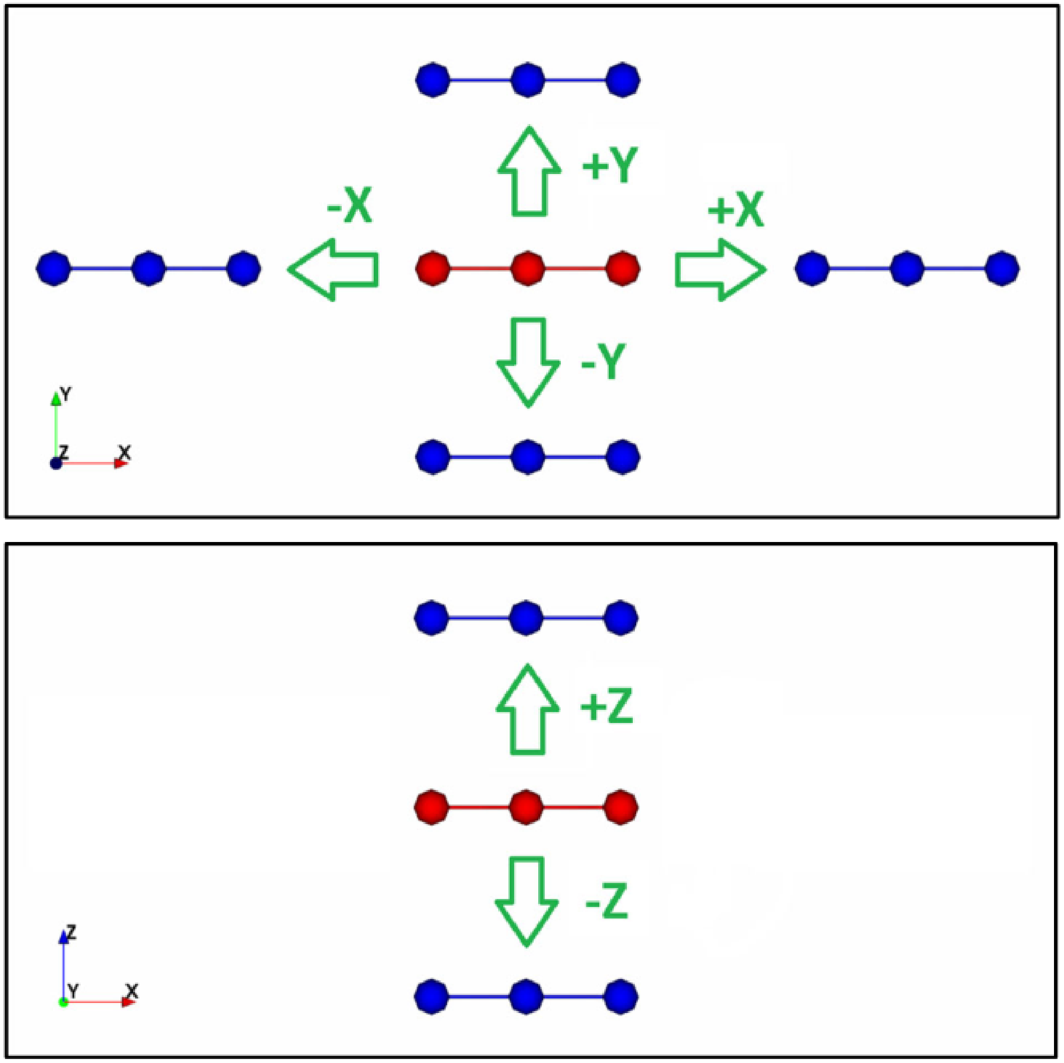


Figure 18.45: An example of the four basic displacement that can be applied along the X/Y axes (A) and along Z axis (B)

The “Move amount” parameter can be used to set the amplitude of the displacement with each mouse click. The default value is 1 (meter), but it can be changed to move the groups of the desired step at once.

18.3 Remove Duplicated

Click with the right mouse button on the node "Electrode", choose "*Tool*", and then *Remove Duplicated* tool (Figure 18.46).

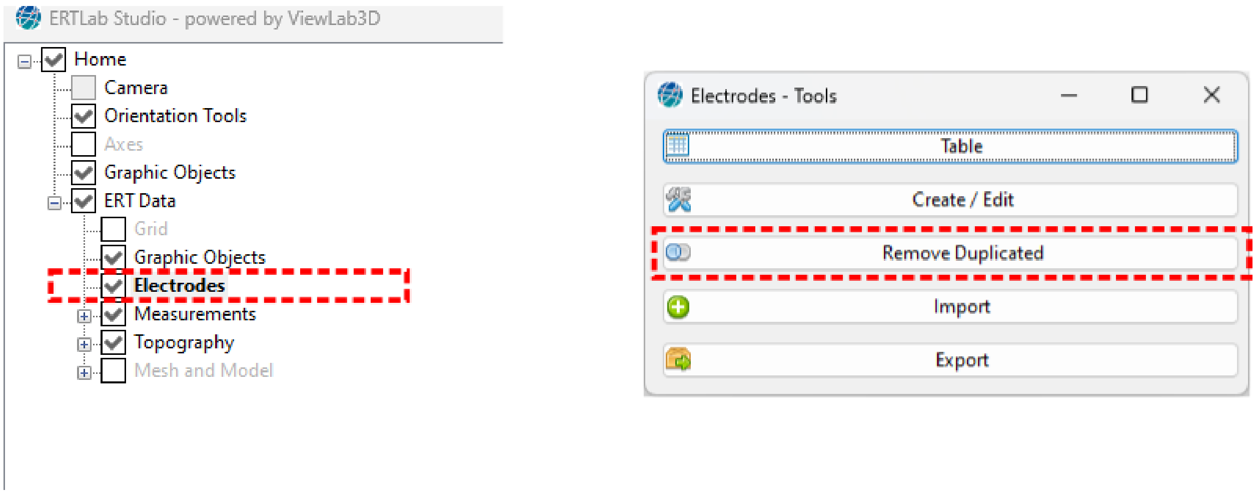


Figure 18.46: Tool Remove Duplicated

Through this tool it is possible to remove duplicate points, which are two or more measurements belonging to the same quadrupole. Duplicate measurements may originate from an electrode recorded by the topographer two times with a small difference or from two 2D lines which intersect each other not in correspondence of the same electrode.

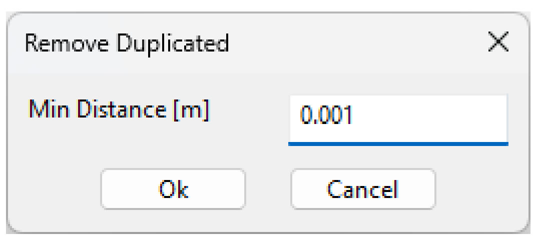


Figure 18.47: Setting of minimum distance for duplicated electrodes

The "*Min Distance*" is the tolerance of the distance between two electrodes. At values higher than those indicated the electrodes are considered as two individual entities and at lower values the two electrodes are merged into a single electrode. At the end of the process a window will inform the user about the number of duplicated quadrupoles removed (Figure 18.48).

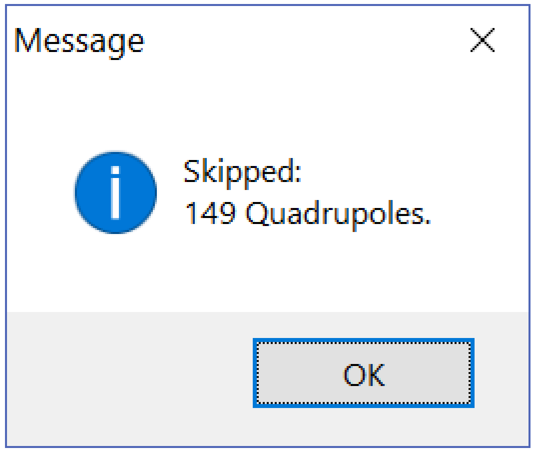


Figure 18.48: Skipped electrode message

18.4 Import

Click with the right mouse button on the node "Electrode", choose "*Tool*", and then *Import* tool (Figure 18.49).

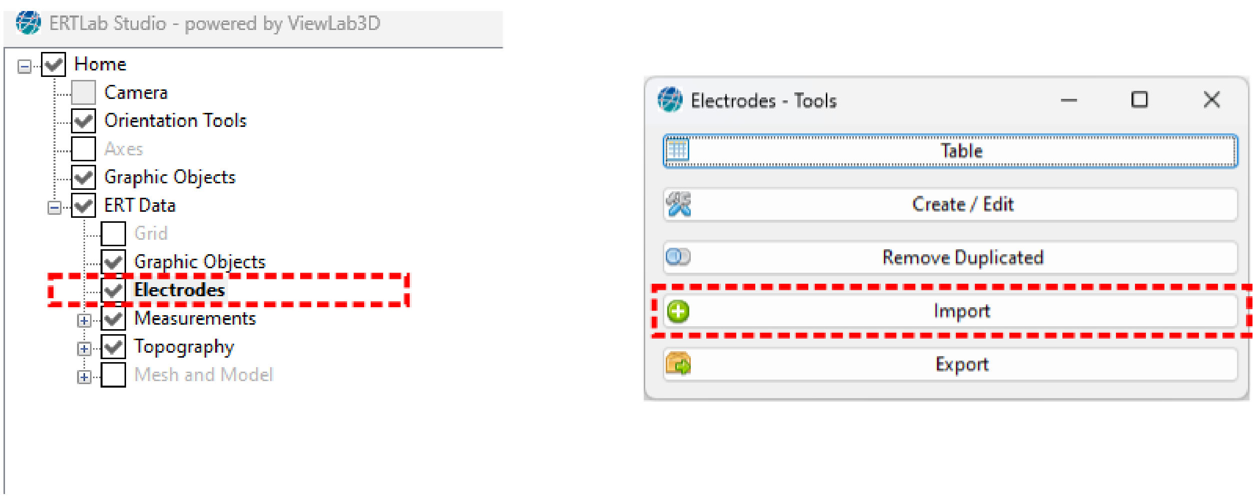


Figure 18.49: Import electrodes tool

Through this tool it is possible to import electrodes. It is possible to load:

- XYZ file (*.xyz; *.txt);
- DATA file (*.data; *.wDat; *.txt);
- Google Earth file (*.KMZ). In this case the coordinates of the electrodes must be in UTM system.

18.5 Export

Click with the right mouse button on the node "Electrode", choose "*Tool*", and then *Export* tool (Figure 18.50).

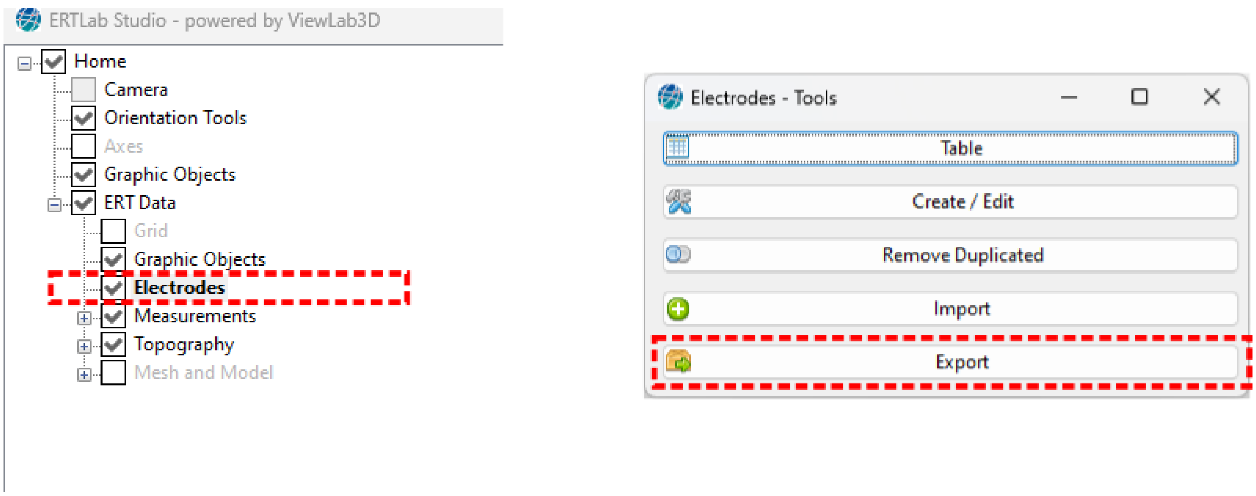


Figure 18.50: Electrode Tools_Eexport

Through this tool it is possible to export a file containing the coordinates of the electrodes used in the loaded project. The exported file is a 3 column .txt file, as shown in Figure 18.51.

Example Data_Electrodes.xyz -

586.01	563.628	314.6
584.01	563.628	314.6
582.01	563.628	314.6
580.01	563.628	314.6
580.01	565.328	314.6
578.01	565.328	314.6
576.01	565.328	314.6
574.012	565.223	314.6
572.015	565.118	314.6
570.018	565.013	314.6

Figure 18.51: Part of electrodes exported file (X,Y,Z coordinates)

CHAPTER

NINETEEN

MEASUREMENTS TOOLS

Right clicking on the node the "*Measurements*" option, a panel will appear with the available tools (Figure 19.1).

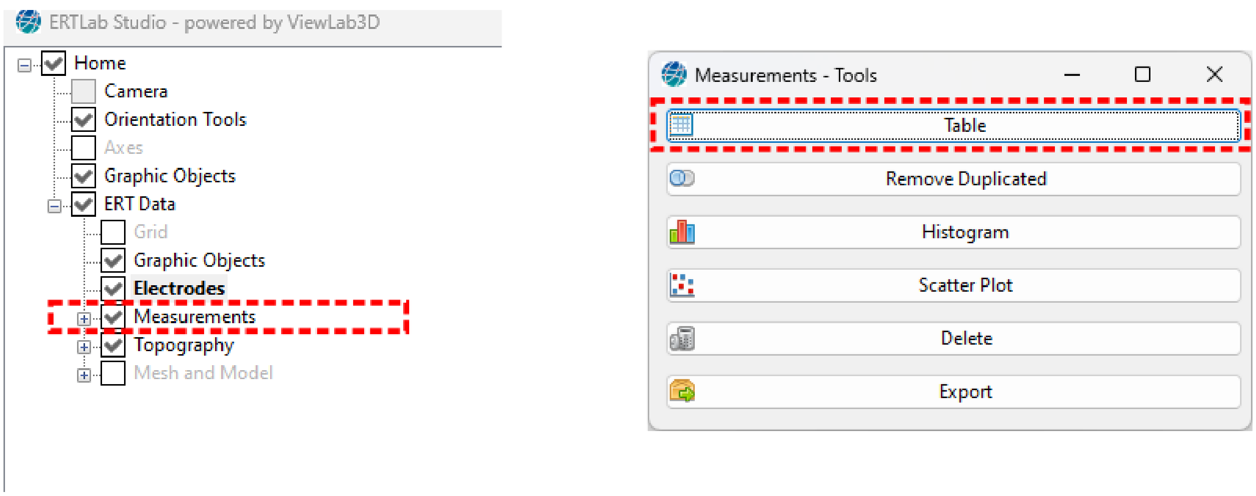


Figure 19.1: Measurements tools panel

All the options are explained in detail in the following sections.

19.1 Table

This table contains all available measurements. In contrast to the electrode table, this table is read only.

Table

ID	Gr_A	El_A	Gr_B	El_B	Gr_M	El_M	Gr_N	El_N	V [mV]	V/I [Ohm]	App Res [Ohm*m]	Current [mA]	IP [mV/V]	StdDev V [V]	StdDev V/I [Ohm]	Skip
1	PD	24	Rem_1	1	PD	48	PD	47	-3.251825808	-0.007252	22.84	448.404	0	0.0002002	0.0004465	false
2	PD	24	Rem_1	1	PD	47	PD	46	-3.553153296	-0.007924	22	448.404	0	0.0002179	0.0004858	false
3	PD	24	Rem_1	1	PD	46	PD	45	-4.355348052	-0.009713	23.65	448.404	0	0.0002651	0.0005913	false
4	PD	24	Rem_1	1	PD	45	PD	44	-5.301928896	-0.011824	25.12	448.404	0	0.0003213	0.0007165	false
5	PD	24	Rem_1	1	PD	44	PD	43	-7.4771367	-0.016675	30.76	448.404	0	0.0004509	0.001005	false
6	PD	24	Rem_1	1	PD	43	PD	42	-7.236343752	-0.016138	25.76	448.404	0	0.0004365	0.0009734	false
7	PD	24	Rem_1	1	PD	42	PD	41	-10.872900192	-0.024248	33.43	448.404	0	0.0006539	0.001458	false
8	PD	24	Rem_1	1	PD	41	PD	40	-12.66517098	-0.028245	33.71	448.404	0	0.0007612	0.001698	false
9	PD	24	Rem_1	1	PD	40	PD	39	-9.4837446	-0.02115	22.05	448.404	0	0.0005708	0.001273	false
10	PD	24	Rem_1	1	PD	39	PD	38	-11.81992944	-0.02636	24.53	448.404	0	0.0007106	0.001585	false
11	PD	24	Rem_1	1	PD	38	PD	37	-13.308592959	-0.029583	25.67	449.873	0	0.0007998	0.001778	false
12	PD	24	Rem_1	1	PD	37	PD	36	-12.163216301	-0.027037	23.82	449.873	0	0.0007312	0.001625	false
13	PD	24	Rem_1	1	PD	36	PD	35	-13.566820061	-0.030157	31.81	449.873	0	0.0008153	0.001812	false
14	PD	24	Rem_1	1	PD	35	PD	34	-6.104326737	-0.013569	25.02	449.873	0	0.000369	0.0008203	false
15	PD	24	Rem_1	1	PD	33	PD	32	15.418497329	0.034273	56.84	449.873	0	0.0009262	0.002059	false
16	PD	24	Rem_1	1	PD	28	PD	27	-78.584265513	-0.174681	22.37	449.873	0	0.004715	0.01048	false
17	PD	24	Rem_1	1	PD	28	PD	26	-274.838355836	-0.608971	24.33	451.316	0	0.01649	0.03654	false
18	PD	24	Rem_1	1	PD	26	PD	25	-491.40188712	-1.08882	15.61	451.316	0	0.02948	0.06533	false
19	PD	24	Rem_1	1	PD	25	PD	27	678.189393988	1.502693	17.28	451.316	0	0.04069	0.09016	false
20	PD	24	Rem_1	1	PD	27	PD	26	-187.85207244	-0.415095	24.1	452.552	0	0.01127	0.02491	false

Figure 19.2: Measurements table

Opening the table shows the following columns:

- ID: Identification Number;
- Gr_A: Name of the group to which the electrode A belongs;
- El_A: Number of electrode A;
- Gr_B: Name of the group to which the electrode B belongs;
- El_B: Number of electrode B;
- Gr_M: Name of the group to which the electrode M belongs;
- El_M: Number of electrode M;
- Gr_N: Name of the group to which the electrode N belongs;
- El_N: Number of electrode N;
- K [m]: Geometrical Factor;
- V [mV]: Electrical Potential;
- V/I [Ohm]: Resistance;
- App Res [Ohm*m]: Apparent Resistivity;
- Current [mA]: Current between El_A and El_B;
- IP [mV/V]: Induced Polarization;
- Standard Deviation V[V]; Standard deviation of Electrical Potential;
- Standard Deviation V/I [Ohm]; Standard deviation of Resistance;
- Skip: if it is “true”, the data is not used for the inversion, but it is not deleted from the dataset memory.

Further options are available right clicking and choosing "*Show All*" (Figure 19.3).

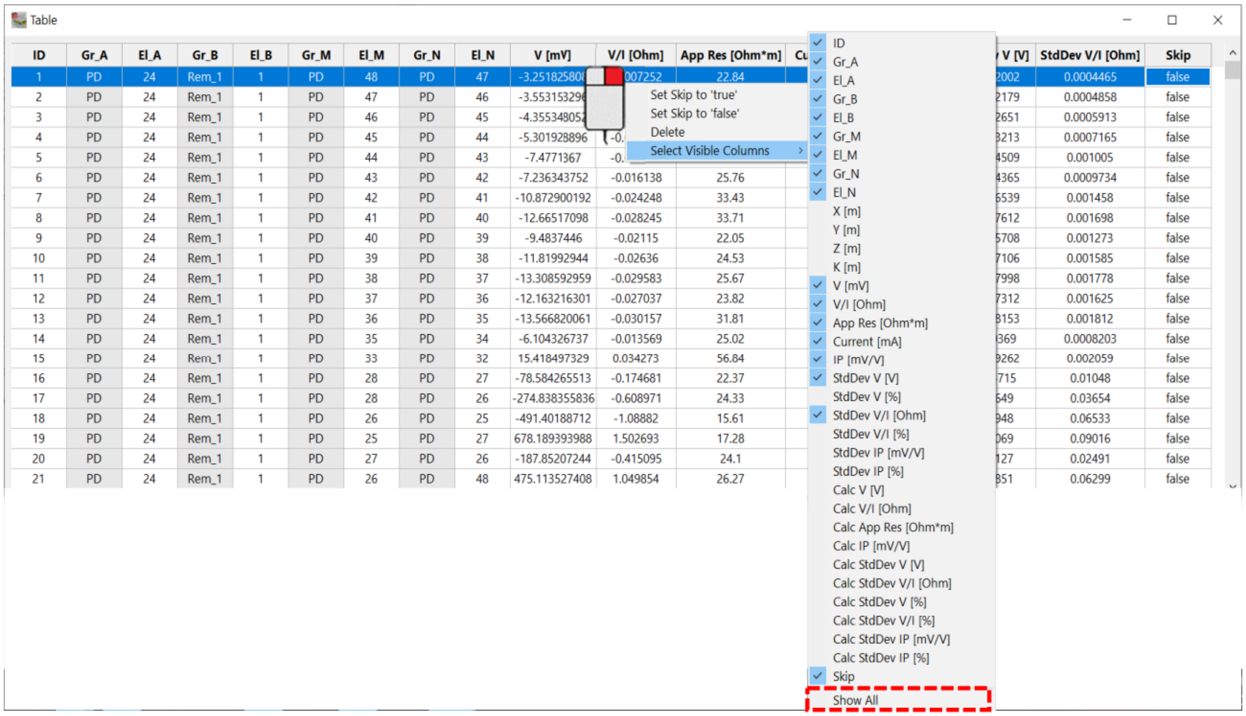


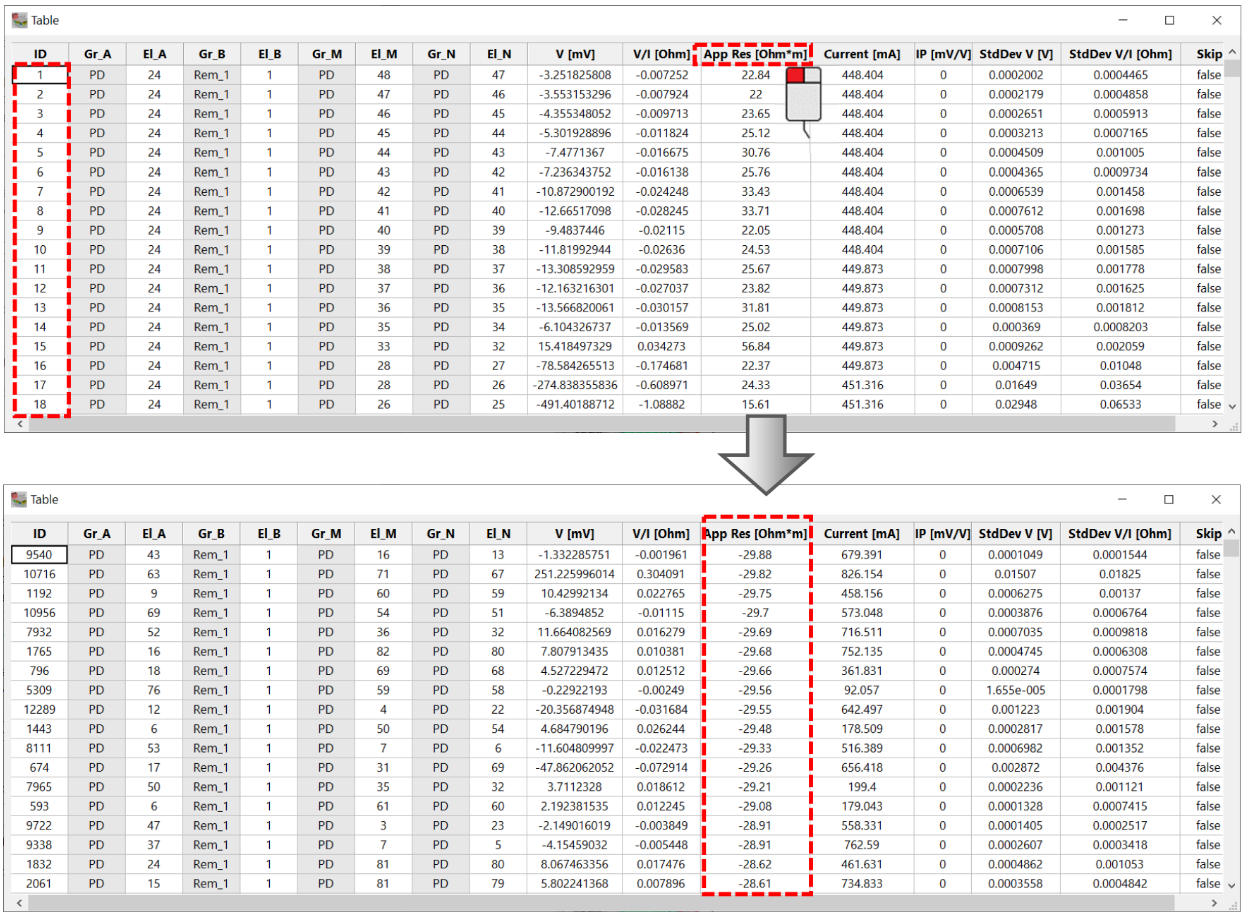
Figure 19.3: Show all visible columns

- X [m]: X Coordinate of the electrode;
- Y [m]: Y Coordinate of the electrode;
- Z [m]: Z Coordinate of the electrode;
- StdDev V [%]: Standard Deviation of Electrical Potential in percentage;
- StdDev V/I [%]: Standard Deviation of Resistance in percentage;
- StdDev IP [mV/V]: Standard Deviation of Induced Potential;
- StdDev IP [%]: Standard Deviation of Induced Potential in percentage;
- Calc V[V]: Post Inversion/Post Forward Model Electrical Potential;
- Calc V/I [Ohm]: Post Inversion/Post Forward Model Resistance;
- Calc App Res [Ohm*m]: Post Inversion/Post Forward Model Apparent Resistivity;
- Calc IP [mV/V]: Post Inversion/Post Forward Model Induced Potential;
- Calc StdDev V[V]: Post Inversion/Post Forward Model Electrical Potential;
- Calc StdDev V/I[Ohm]: Post Inversion/Post Forward Model Resistance;
- Calc StdDev V [%]: Post Inversion/Post Forward Model Electrical Potential in percentage;

- Calc StdDev V/I [%]: Post Inversion/Post Forward Model Resistance in percentage;
- Calc StdDev IP [mV/V]: Post Inversion/Post Forward Model Induced Polarization;
- Calc StdDev IP [%]: Post Inversion/Post Forward Model Induced Polarization in percentage.

Left clicking on one of the column headings sorts all data in increasing order of that column. Left clicking a second times on the same column will sort the data in decreasing order.

Figure 19.4 shows an example where data are sorted by increasing Apparent Resistivity.



Table

ID	Gr_A	El_A	Gr_B	El_B	Gr_M	El_M	Gr_N	El_N	V [mV]	V/I [Ohm]	App Res [Ohm*m]	Current [mA]	IP [mV/V]	StdDev V [V]	StdDev V/I [Ohm]	Skip
1	PD	24	Rem_1	1	PD	48	PD	47	-3.251825808	-0.007252	22.84	448.404	0	0.0002002	0.0004465	false
2	PD	24	Rem_1	1	PD	47	PD	46	-3.553153296	-0.007924	22	448.404	0	0.0002179	0.0004858	false
3	PD	24	Rem_1	1	PD	46	PD	45	-4.355348052	-0.009713	23.65	448.404	0	0.0002651	0.0005913	false
4	PD	24	Rem_1	1	PD	45	PD	44	-5.301928896	-0.011824	25.12	448.404	0	0.0003213	0.0007165	false
5	PD	24	Rem_1	1	PD	44	PD	43	-7.4771367	-0.016675	30.76	448.404	0	0.0004509	0.001005	false
6	PD	24	Rem_1	1	PD	43	PD	42	-7.236343752	-0.016138	25.76	448.404	0	0.0004365	0.0009734	false
7	PD	24	Rem_1	1	PD	42	PD	41	-10.872900192	-0.024248	33.43	448.404	0	0.0006539	0.001458	false
8	PD	24	Rem_1	1	PD	41	PD	40	-12.66517098	-0.028245	33.71	448.404	0	0.0007612	0.001698	false
9	PD	24	Rem_1	1	PD	40	PD	39	-9.4837446	-0.02115	22.05	448.404	0	0.0005708	0.001273	false
10	PD	24	Rem_1	1	PD	39	PD	38	-11.81992944	-0.02636	24.53	448.404	0	0.0007106	0.001585	false
11	PD	24	Rem_1	1	PD	38	PD	37	-13.308592959	-0.029583	25.67	449.873	0	0.0007998	0.001778	false
12	PD	24	Rem_1	1	PD	37	PD	36	-12.163216301	-0.027037	23.82	449.873	0	0.0007312	0.001625	false
13	PD	24	Rem_1	1	PD	36	PD	35	-13.566820061	-0.030157	31.81	449.873	0	0.0008153	0.001812	false
14	PD	24	Rem_1	1	PD	35	PD	34	-6.104326737	-0.013569	25.02	449.873	0	0.000369	0.0008203	false
15	PD	24	Rem_1	1	PD	33	PD	32	15.418497329	0.034273	56.84	449.873	0	0.0009262	0.002059	false
16	PD	24	Rem_1	1	PD	28	PD	27	-78.584265513	-0.174681	22.37	449.873	0	0.004715	0.01048	false
17	PD	24	Rem_1	1	PD	28	PD	26	-274.838355836	-0.608971	24.33	451.316	0	0.01649	0.03654	false
18	PD	24	Rem_1	1	PD	26	PD	25	-491.40188712	-1.08882	15.61	451.316	0	0.02948	0.06533	false

Table

ID	Gr_A	El_A	Gr_B	El_B	Gr_M	El_M	Gr_N	El_N	V [mV]	V/I [Ohm]	App Res [Ohm*m]	Current [mA]	IP [mV/V]	StdDev V [V]	StdDev V/I [Ohm]	Skip
9540	PD	43	Rem_1	1	PD	16	PD	13	-1.332285751	-0.001961	-29.88	679.391	0	0.0001049	0.0001544	false
10716	PD	63	Rem_1	1	PD	71	PD	67	251.225996014	0.304091	-29.82	826.154	0	0.01507	0.01825	false
1192	PD	9	Rem_1	1	PD	60	PD	59	10.42992134	0.022765	-29.75	458.156	0	0.0006275	0.00137	false
10956	PD	69	Rem_1	1	PD	54	PD	51	-6.3894852	-0.01115	-29.7	573.048	0	0.0003876	0.0006764	false
7932	PD	52	Rem_1	1	PD	36	PD	32	11.664082569	0.016279	-29.69	716.511	0	0.0007035	0.0009818	false
1765	PD	16	Rem_1	1	PD	82	PD	80	7.807913435	0.010381	-29.68	752.135	0	0.0004745	0.0006308	false
796	PD	18	Rem_1	1	PD	69	PD	68	4.527229472	0.012512	-29.66	361.831	0	0.000274	0.0007574	false
5309	PD	76	Rem_1	1	PD	59	PD	58	-0.22922193	-0.00249	-29.56	92.057	0	1.655e-005	0.0001798	false
12289	PD	12	Rem_1	1	PD	4	PD	22	-20.356874948	-0.031684	-29.55	642.497	0	0.001223	0.001904	false
1443	PD	6	Rem_1	1	PD	50	PD	54	4.684790196	0.026244	-29.48	178.509	0	0.0002817	0.001578	false
8111	PD	53	Rem_1	1	PD	7	PD	6	-11.604809997	-0.022473	-29.33	516.389	0	0.0006982	0.001352	false
674	PD	17	Rem_1	1	PD	31	PD	69	-47.862062052	-0.072914	-29.26	656.418	0	0.002872	0.004376	false
7965	PD	50	Rem_1	1	PD	35	PD	32	3.7112328	0.018612	-29.21	199.4	0	0.0002236	0.001121	false
593	PD	6	Rem_1	1	PD	61	PD	60	2.192381535	0.012245	-29.08	179.043	0	0.0001328	0.0007415	false
9722	PD	47	Rem_1	1	PD	3	PD	23	-2.149016019	-0.003849	-28.91	558.331	0	0.0001405	0.0002517	false
9338	PD	37	Rem_1	1	PD	7	PD	5	-4.15459032	-0.005448	-28.91	762.59	0	0.0002607	0.0003418	false
1832	PD	24	Rem_1	1	PD	81	PD	80	8.067463356	0.017476	-28.62	461.631	0	0.0004862	0.001053	false
2061	PD	15	Rem_1	1	PD	81	PD	79	5.802241368	0.007896	-28.61	734.833	0	0.0003558	0.0004842	false

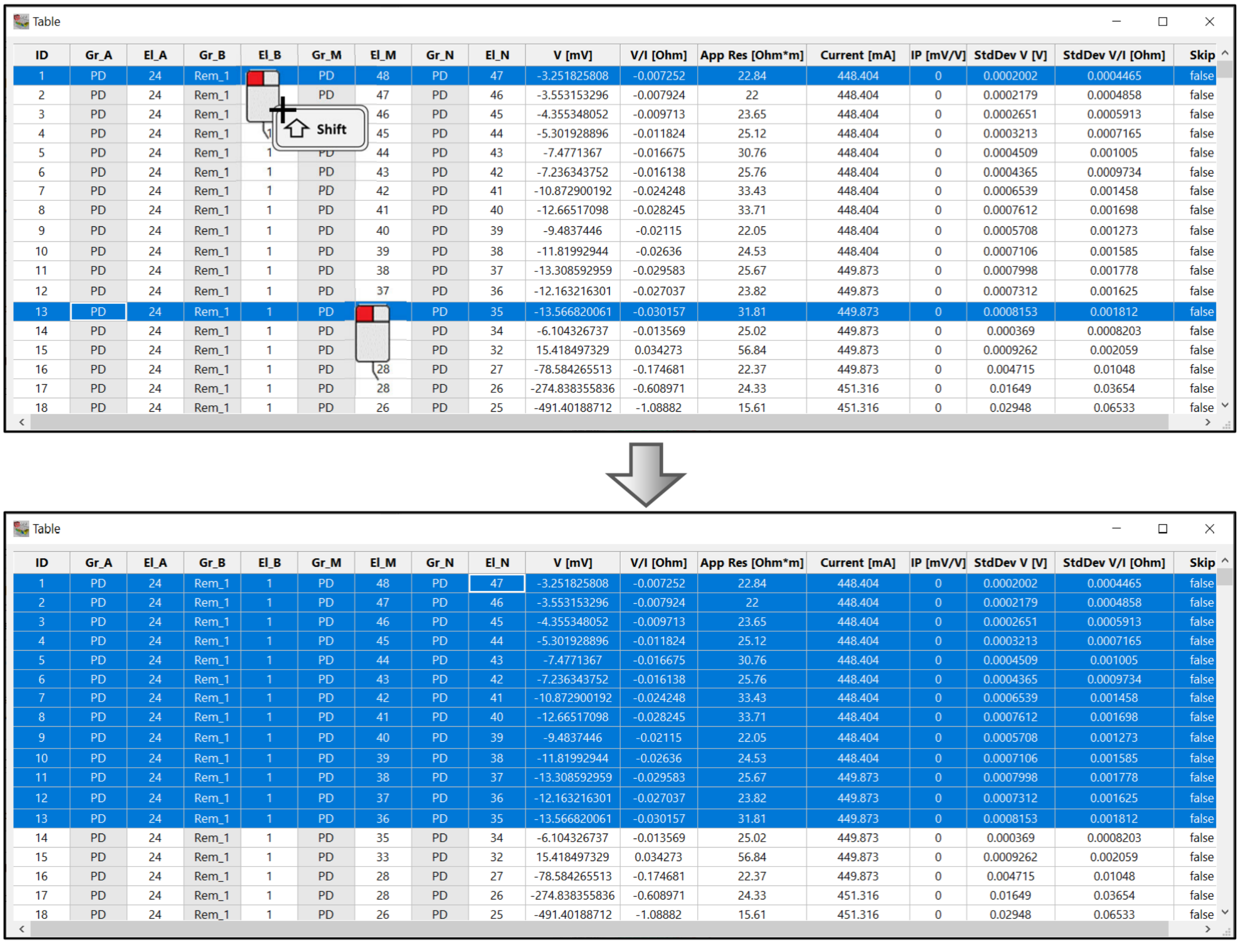
Figure 19.4: Sorting of Measurements Table. By default, it is sorted by increasing ID

Right clicking on any point of the table allows the user to:

- **Set skip to true**: in this way the quadrupole is not used for the inversion, even if it is not deleted from the data file;
- **Set skip to false**: in this way the quadrupole is used for the inversion; by default, all quadrupoles are set to "false";
- **Delete**: deletes the selected data from the project.

It is possible to do multiple selections by left clicking in the first and last row of the wanted selection and holding the SHIFT button all the lines between clicking (Figure 19.5).

To Skip (or unkip) a measurement it is also possible to simply double click the related row in the table.

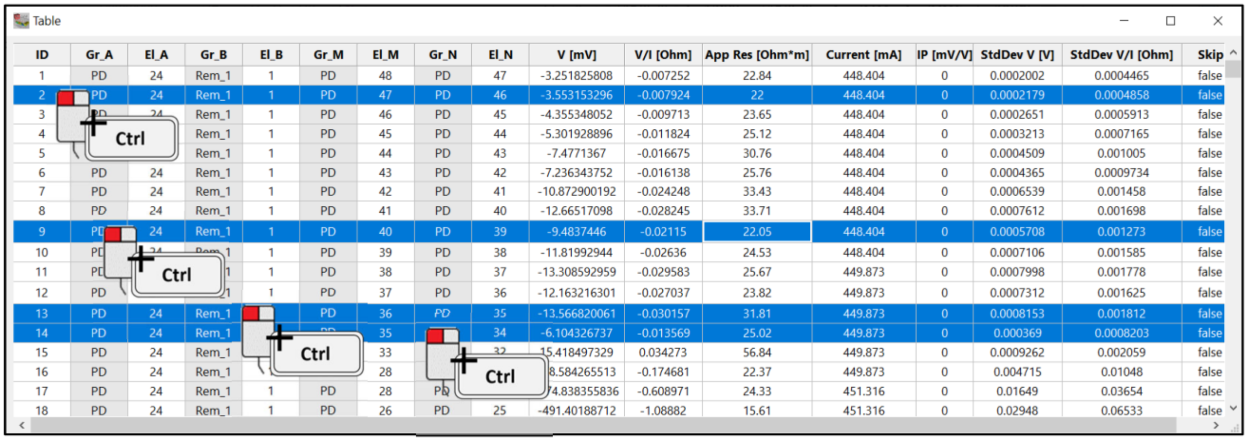


ID	Gr_A	El_A	Gr_B	El_B	Gr_M	El_M	Gr_N	El_N	V [mV]	V/I [Ohm]	App Res [Ohm*m]	Current [mA]	IP [mV/V]	StdDev V [V]	StdDev V/I [Ohm]	Skip
1	PD	24	Rem_1	[illegible]	PD	48	PD	47	-3.251825808	-0.007252	22.84	448.404	0	0.0002002	0.0004465	false
2	PD	24	Rem_1	[illegible]	PD	47	PD	46	-3.553153296	-0.007924	22	448.404	0	0.0002179	0.0004858	false
3	PD	24	Rem_1	[illegible]	[illegible]	46	PD	45	-4.355348052	-0.009713	23.65	448.404	0	0.0002651	0.0005913	false
4	PD	24	Rem_1	[illegible]	[illegible]	45	PD	44	-5.301928896	-0.011824	25.12	448.404	0	0.0003213	0.0007165	false
5	PD	24	Rem_1	1	PD	44	PD	43	-7.4771367	-0.016675	30.76	448.404	0	0.0004509	0.001005	false
6	PD	24	Rem_1	1	PD	43	PD	42	-7.236343752	-0.016138	25.76	448.404	0	0.0004365	0.0009734	false
7	PD	24	Rem_1	1	PD	42	PD	41	-10.872900192	-0.024248	33.43	448.404	0	0.0006539	0.001458	false
8	PD	24	Rem_1	1	PD	41	PD	40	-12.66517098	-0.028245	33.71	448.404	0	0.0007612	0.001698	false
9	PD	24	Rem_1	1	PD	40	PD	39	-9.4837446	-0.02115	22.05	448.404	0	0.0005708	0.001273	false
10	PD	24	Rem_1	1	PD	39	PD	38	-11.81992944	-0.02636	24.53	448.404	0	0.0007106	0.001585	false
11	PD	24	Rem_1	1	PD	38	PD	37	-13.308592959	-0.029583	25.67	449.873	0	0.0007998	0.001778	false
12	PD	24	Rem_1	1	PD	37	PD	36	-12.163216301	-0.027037	23.82	449.873	0	0.0007312	0.001625	false
13	PD	24	Rem_1	1	PD	[illegible]	PD	35	-13.566820061	-0.030157	31.81	449.873	0	0.0008153	0.001812	false
14	PD	24	Rem_1	1	PD	[illegible]	PD	34	-6.104326737	-0.013569	25.02	449.873	0	0.000369	0.0008203	false
15	PD	24	Rem_1	1	PD	[illegible]	PD	32	15.418497329	0.034273	56.84	449.873	0	0.0009262	0.002059	false
16	PD	24	Rem_1	1	PD	28	PD	27	-78.584265513	-0.174681	22.37	449.873	0	0.004715	0.01048	false
17	PD	24	Rem_1	1	PD	28	PD	26	-274.838355836	-0.608971	24.33	451.316	0	0.01649	0.03654	false
18	PD	24	Rem_1	1	PD	26	PD	25	-491.40188712	-1.08882	15.61	451.316	0	0.02948	0.06533	false

ID	Gr_A	El_A	Gr_B	El_B	Gr_M	El_M	Gr_N	El_N	V [mV]	V/I [Ohm]	App Res [Ohm*m]	Current [mA]	IP [mV/V]	StdDev V [V]	StdDev V/I [Ohm]	Skip
1	PD	24	Rem_1	1	PD	48	PD	47	-3.251825808	-0.007252	22.84	448.404	0	0.0002002	0.0004465	false
2	PD	24	Rem_1	1	PD	47	PD	46	-3.553153296	-0.007924	22	448.404	0	0.0002179	0.0004858	false
3	PD	24	Rem_1	1	PD	46	PD	45	-4.355348052	-0.009713	23.65	448.404	0	0.0002651	0.0005913	false
4	PD	24	Rem_1	1	PD	45	PD	44	-5.301928896	-0.011824	25.12	448.404	0	0.0003213	0.0007165	false
5	PD	24	Rem_1	1	PD	44	PD	43	-7.4771367	-0.016675	30.76	448.404	0	0.0004509	0.001005	false
6	PD	24	Rem_1	1	PD	43	PD	42	-7.236343752	-0.016138	25.76	448.404	0	0.0004365	0.0009734	false
7	PD	24	Rem_1	1	PD	42	PD	41	-10.872900192	-0.024248	33.43	448.404	0	0.0006539	0.001458	false
8	PD	24	Rem_1	1	PD	41	PD	40	-12.66517098	-0.028245	33.71	448.404	0	0.0007612	0.001698	false
9	PD	24	Rem_1	1	PD	40	PD	39	-9.4837446	-0.02115	22.05	448.404	0	0.0005708	0.001273	false
10	PD	24	Rem_1	1	PD	39	PD	38	-11.81992944	-0.02636	24.53	448.404	0	0.0007106	0.001585	false
11	PD	24	Rem_1	1	PD	38	PD	37	-13.308592959	-0.029583	25.67	449.873	0	0.0007998	0.001778	false
12	PD	24	Rem_1	1	PD	37	PD	36	-12.163216301	-0.027037	23.82	449.873	0	0.0007312	0.001625	false
13	PD	24	Rem_1	1	PD	36	PD	35	-13.566820061	-0.030157	31.81	449.873	0	0.0008153	0.001812	false
14	PD	24	Rem_1	1	PD	35	PD	34	-6.104326737	-0.013569	25.02	449.873	0	0.000369	0.0008203	false
15	PD	24	Rem_1	1	PD	33	PD	32	15.418497329	0.034273	56.84	449.873	0	0.0009262	0.002059	false
16	PD	24	Rem_1	1	PD	28	PD	27	-78.584265513	-0.174681	22.37	449.873	0	0.004715	0.01048	false
17	PD	24	Rem_1	1	PD	28	PD	26	-274.838355836	-0.608971	24.33	451.316	0	0.01649	0.03654	false
18	PD	24	Rem_1	1	PD	26	PD	25	-491.40188712	-1.08882	15.61	451.316	0	0.02948	0.06533	false

Figure 19.5: Sequential selection

Otherwise, to select more than one line, but in a non-sequential order, click on the desired lines and hold the Ctrl button (Figure 19.6):



Table

ID	Gr_A	El_A	Gr_B	El_B	Gr_M	El_M	Gr_N	El_N	V [mV]	V/I [Ohm]	App Res [Ohm*m]	Current [mA]	IP [mV/V]	StdDev V [V]	StdDev V/I [Ohm]	Skip
1	PD	24	Rem_1	1	PD	48	PD	47	-3.251825808	-0.007252	22.84	448.404	0	0.0002002	0.0004465	false
2	PD	24	Rem_1	1	PD	47	PD	46	-3.553153296	-0.007924	22	448.404	0	0.0002179	0.0004858	false
3	[illegible]	[illegible]	Rem_1	1	PD	46	PD	45	-4.355348052	-0.009713	23.65	448.404	0	0.0002651	0.0005913	false
4	[illegible]	[illegible]	Rem_1	1	PD	45	PD	44	-5.301928896	-0.011824	25.12	448.404	0	0.0003213	0.0007165	false
5	[illegible]	[illegible]	Rem_1	1	PD	44	PD	43	-7.4771367	-0.016675	30.76	448.404	0	0.0004509	0.001005	false
6	PD	24	Rem_1	1	PD	43	PD	42	-7.236343752	-0.016138	25.76	448.404	0	0.0004365	0.0009734	false
7	PD	24	Rem_1	1	PD	42	PD	41	-10.872900192	-0.024248	33.43	448.404	0	0.0006539	0.001458	false
8	PD	24	Rem_1	1	PD	41	PD	40	-12.66517098	-0.028245	33.71	448.404	0	0.0007612	0.001698	false
9	[illegible]	24	Rem_1	1	PD	40	PD	39	-9.4837446	-0.02115	22.05	448.404	0	0.0005708	0.001273	false
10	[illegible]	[illegible]	[illegible]	1	PD	39	PD	38	-11.81992944	-0.02636	24.53	448.404	0	0.0007106	0.001585	false
11	[illegible]	[illegible]	[illegible]	1	PD	38	PD	37	-13.308592959	-0.029583	25.67	449.873	0	0.0007998	0.001778	false
12	PD	[illegible]	[illegible]	1	PD	37	PD	36	-12.163216301	-0.027037	23.82	449.873	0	0.0007312	0.001625	false
13	PD	24	Rem_1	[illegible]	PD	36	PD	35	-13.566820061	-0.030157	31.81	449.873	0	0.0008153	0.001812	false
14	PD	24	Rem_1	[illegible]	[illegible]	35	[illegible]	34	-6.104326737	-0.013569	25.02	449.873	0	0.000369	0.0008203	false
15	PD	24	Rem_1	[illegible]	[illegible]	33	[illegible]	[illegible]	[illegible]	0.034273	56.84	449.873	0	0.0009262	0.002059	false
16	PD	24	Rem_1	[illegible]	[illegible]	28	[illegible]	[illegible]	[illegible]	-0.174681	22.37	449.873	0	0.004715	0.01048	false
17	PD	24	Rem_1	1	PD	28	[illegible]	[illegible]	[illegible]	-0.608971	24.33	451.316	0	0.01649	0.03654	false
18	PD	24	Rem_1	1	PD	26	PD	25	-491.40188712	-1.08882	15.61	451.316	0	0.02948	0.06533	false

Figure 19.6: Non-sequential selection

19.2 Remove Duplicated

This tool allows the removal of duplicate and reciprocal measurements.

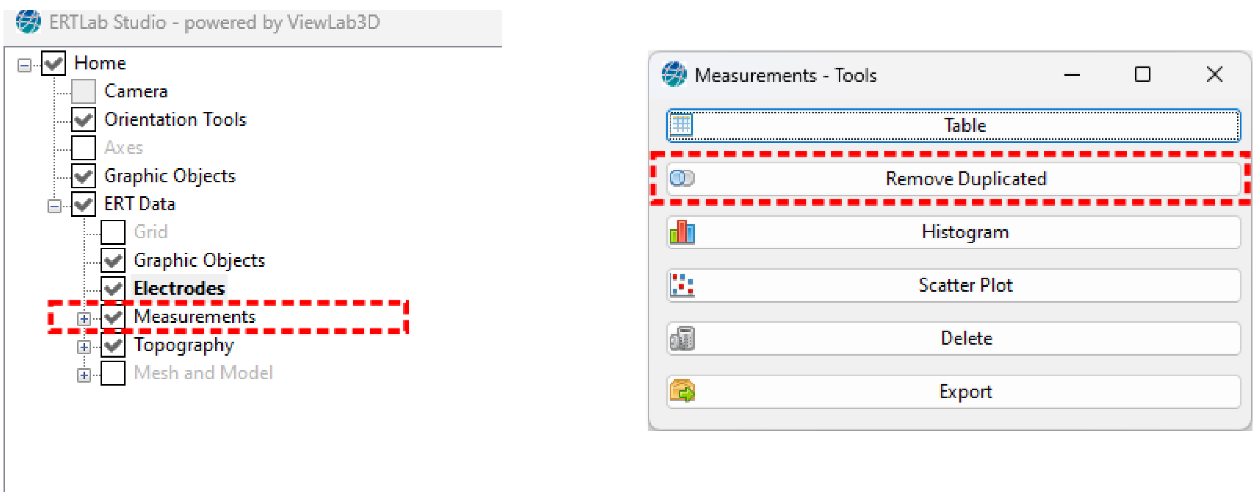
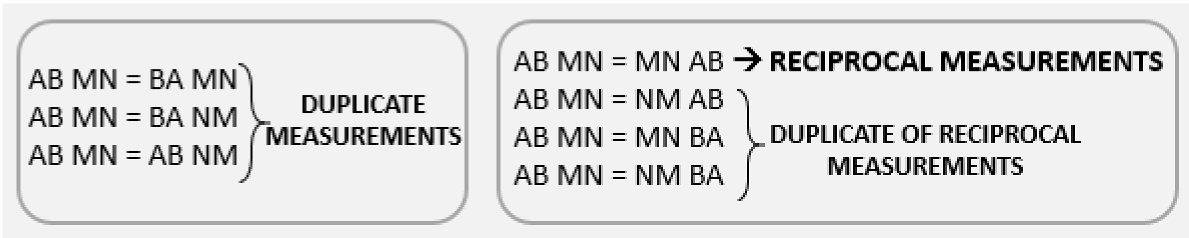


Figure 19.7: Remove duplicated Panel

Reciprocal Measurements have Transmitters (Tx, AB) and Receivers (Rx, MN) reversed. Theoretically, they should give the same measure. Schematically, in a quadrupoles AB_MN the following combinations are possible:



After clicking on the *Remove Duplicated* button and at the end of the computation, a message window will appear with information about the number of reciprocal quadrupoles and the difference of percentage average of Apparent Resistivity (Figure 19.8).

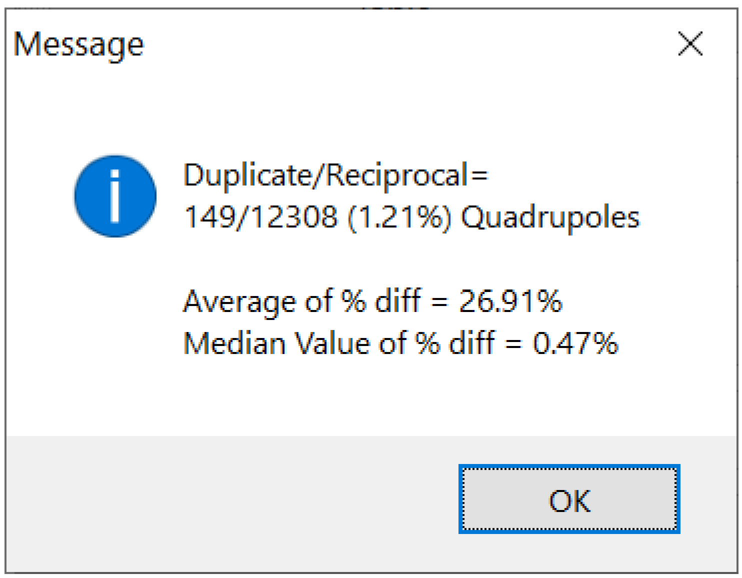


Figure 19.8: Duplicate/Reciprocal message

After clicking on Ok button, a new window will open, providing the user with three choices on what to do with the duplicated/reciprocal measurements. Options are:

- **Export Report to CSV**: exports a .CSV file with information about duplicate/reciprocal measurements.
- **Set as Noise**: the duplicate/reciprocal measurements are attributed to the Rho noise % in the inversion (section Data Error).
- **Remove Duplicated**: deletes the duplicate/reciprocal measurements from the dataset.
- **Keep only Duplicated**: to remove all measurement from the dataset that is not a duplicate/reciprocal measurements.
- **Remove only bad measurement**: to remove duplicate/reciprocal measurement from the dataset that has an high deviation.

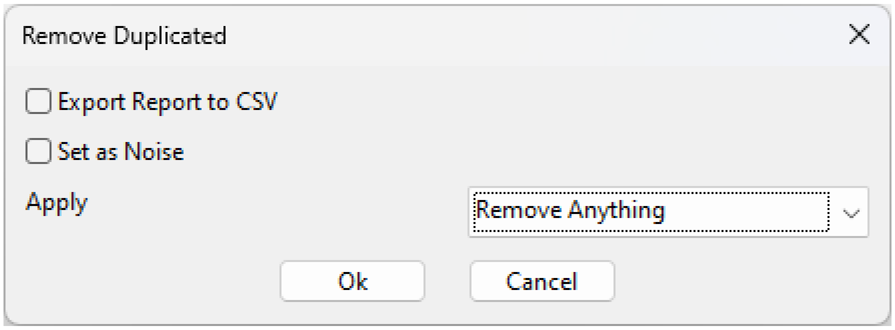


Figure 19.9: Remove Duplicated Measurements

19.3 Histogram

With this tool it is possible to visualize the measurements as Histograms, to have an easier representation of data distribution.

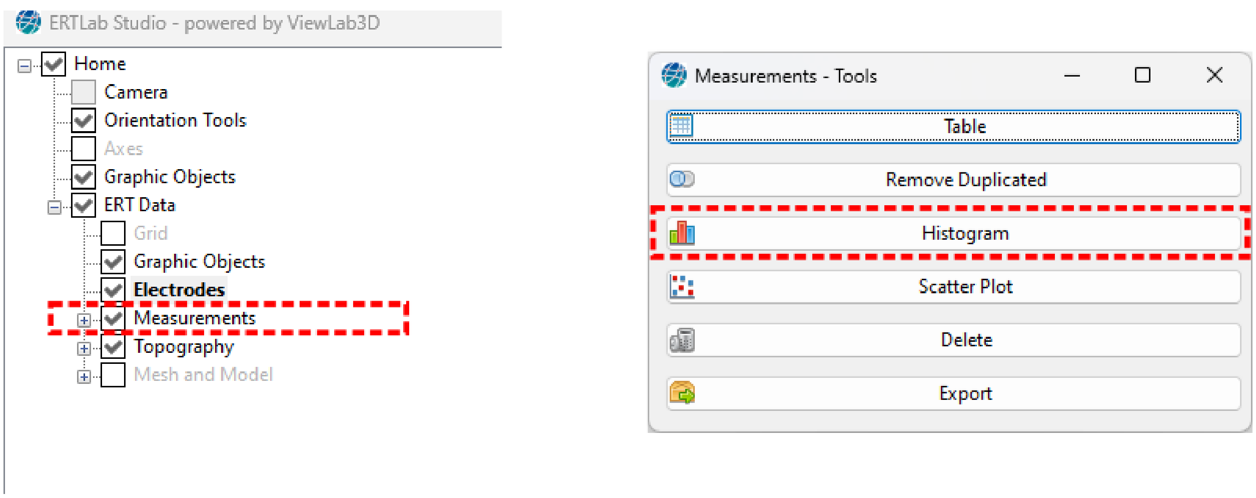


Figure 19.10: Histogram panel

This window has the tools to manage and visualize the data as histograms.

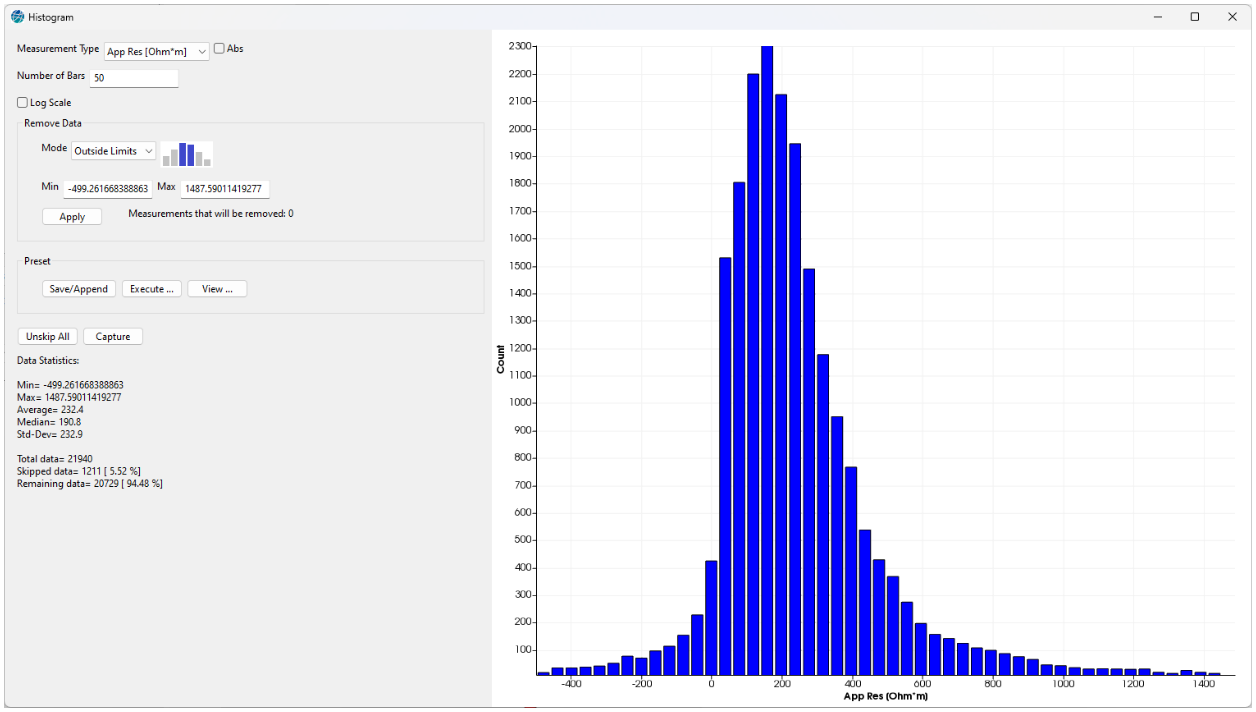


Figure 19.11: Histogram panel

19.3.1 Measurement Type

Choose the type of measurement, to be visualize in a histogram:

- V/I [Ohm]
- Apparent Resistivity [Ohm*m]
- IP [mV/V]
- K [m]
- V [mV]
- I [mA]
- Standard Deviation V/I [Ohm]
- Standard Deviation V[V]
- Standard Deviation IP [mV/V]
- Standard Deviation IP [%]
- Calculated V/I [Ohm]
- Calculated Apparent Resistivity [Ohm*m]
- Calculated IP [mV/V]

- Calculated Standard Deviation V/I [Ohm]
- Calculated Standard Deviation IP [mV/V]
- Calculated Standard Deviation IP [%]

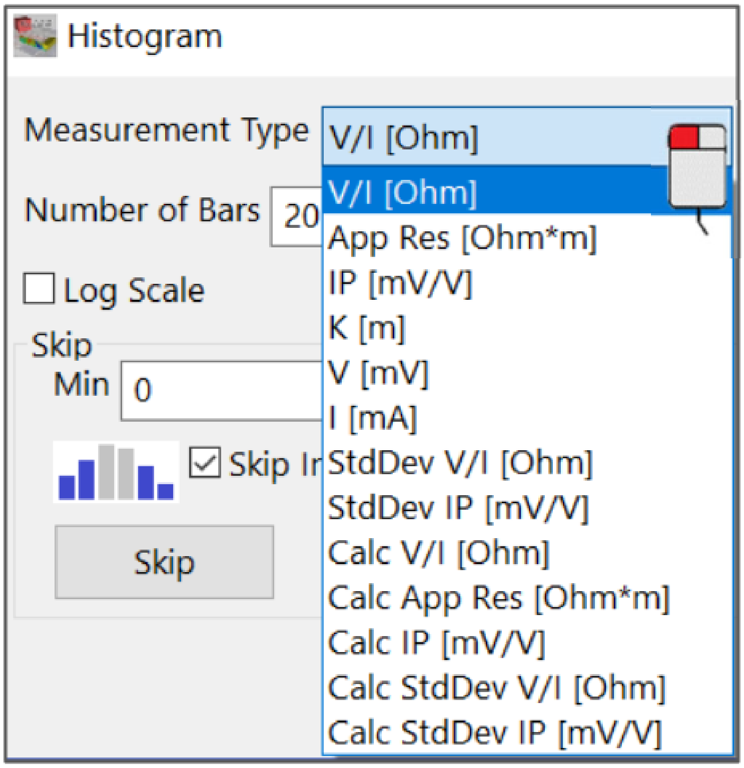


Figure 19.12: Choice of measurements to display as histogram

19.3.2 Number of Bars

The user has the option to change the *number of bars*' for the histogram and set the *logarithmic display* mode for a better data distribution representation. In the following examples the histogram represent the V/I measurement with linear display mode (Figure 19.13 A), logarithmic display mode with 20 bars (Figure 19.13 B) and logarithmic display mode with 50 bars (Figure 19.13 C).

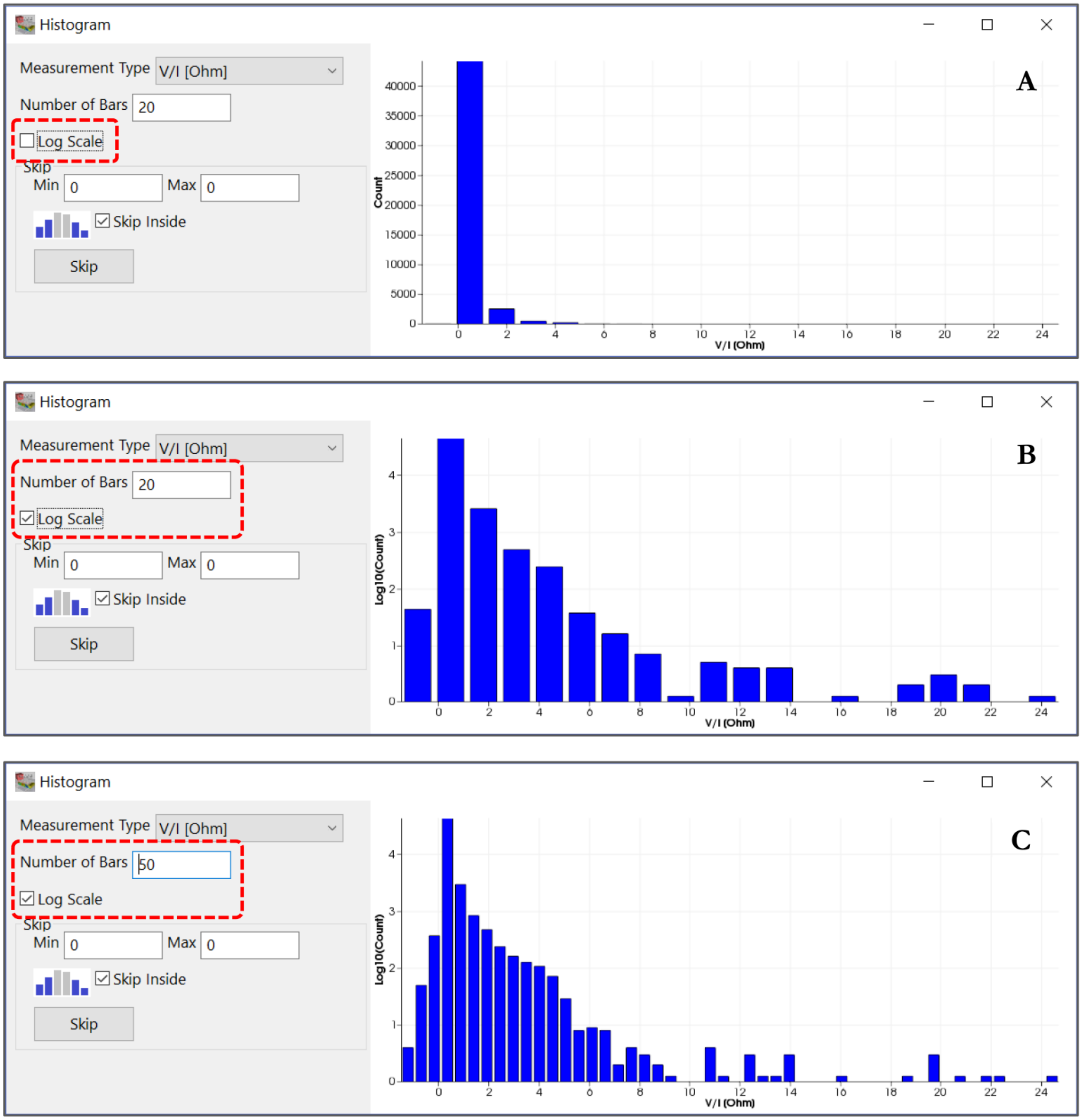


Figure 19.13: Logarithmic and number of bars setting

19.3.3 Mode

Select *outside limits* to filter the data externally to the selected cut limit values and preserve the internal data. Otherwise, by selecting *inside limits* data inside the cut limits are filtered. In Figure 19.14 the data range is set to -1 as minimum value and to 9.5 as maximum value. In the first case the measurements outside the limits are skipped, in the second case those internal.

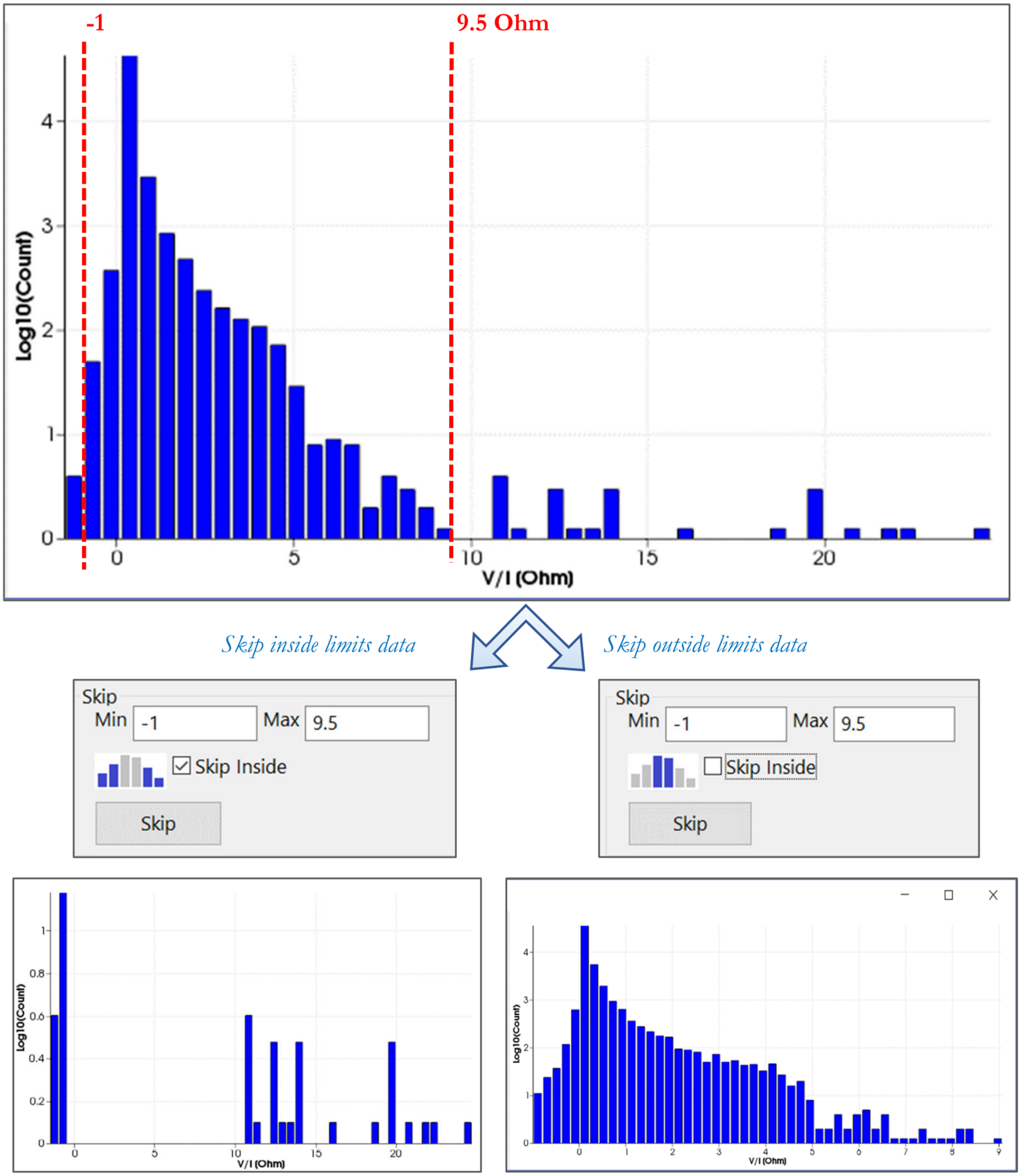


Figure 19.14: Filtering Outside/Inside limits data

19.3.4 Min Max

Examine the amount of filtered data after the application of your filter choices in the *Statistical Summary* panel, where the Minimum and Maximum values of the dataset, the Average and Median values, the total number of measurements, and the number of skipped data are reported.

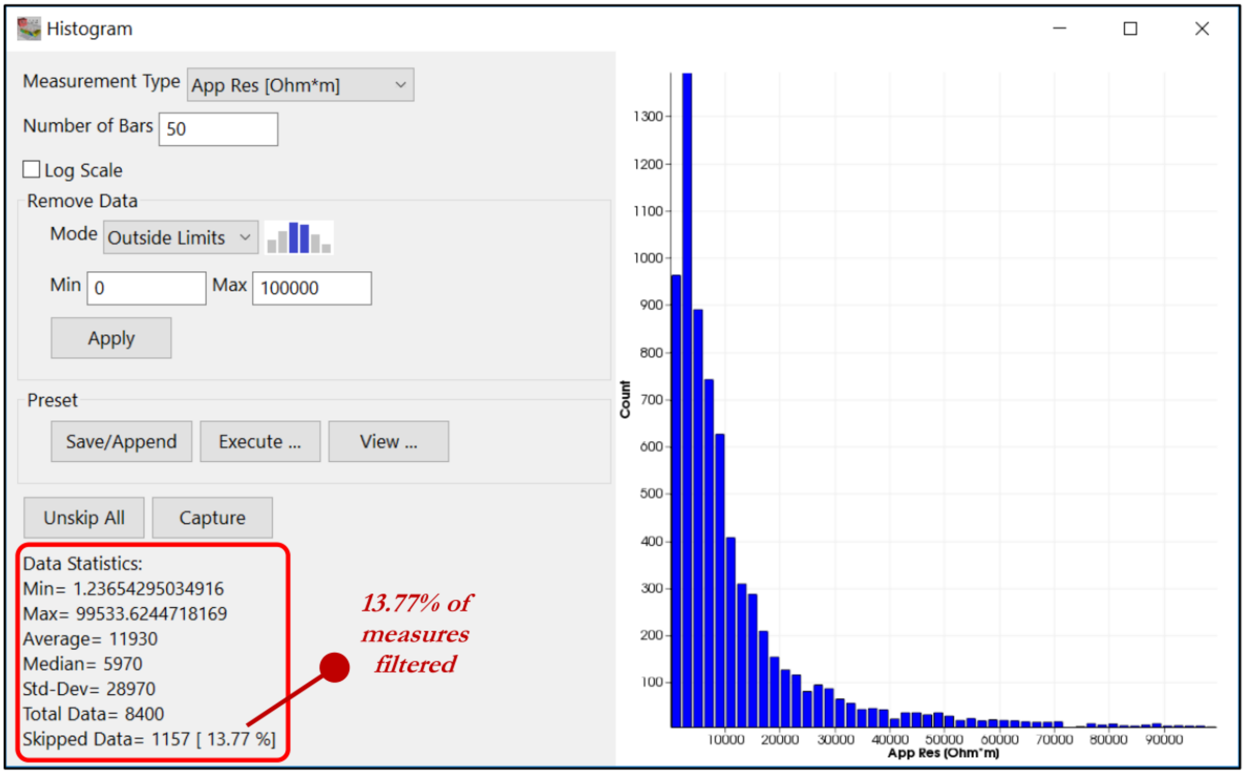


Figure 19.15: Data Statistics in Histogram Panel

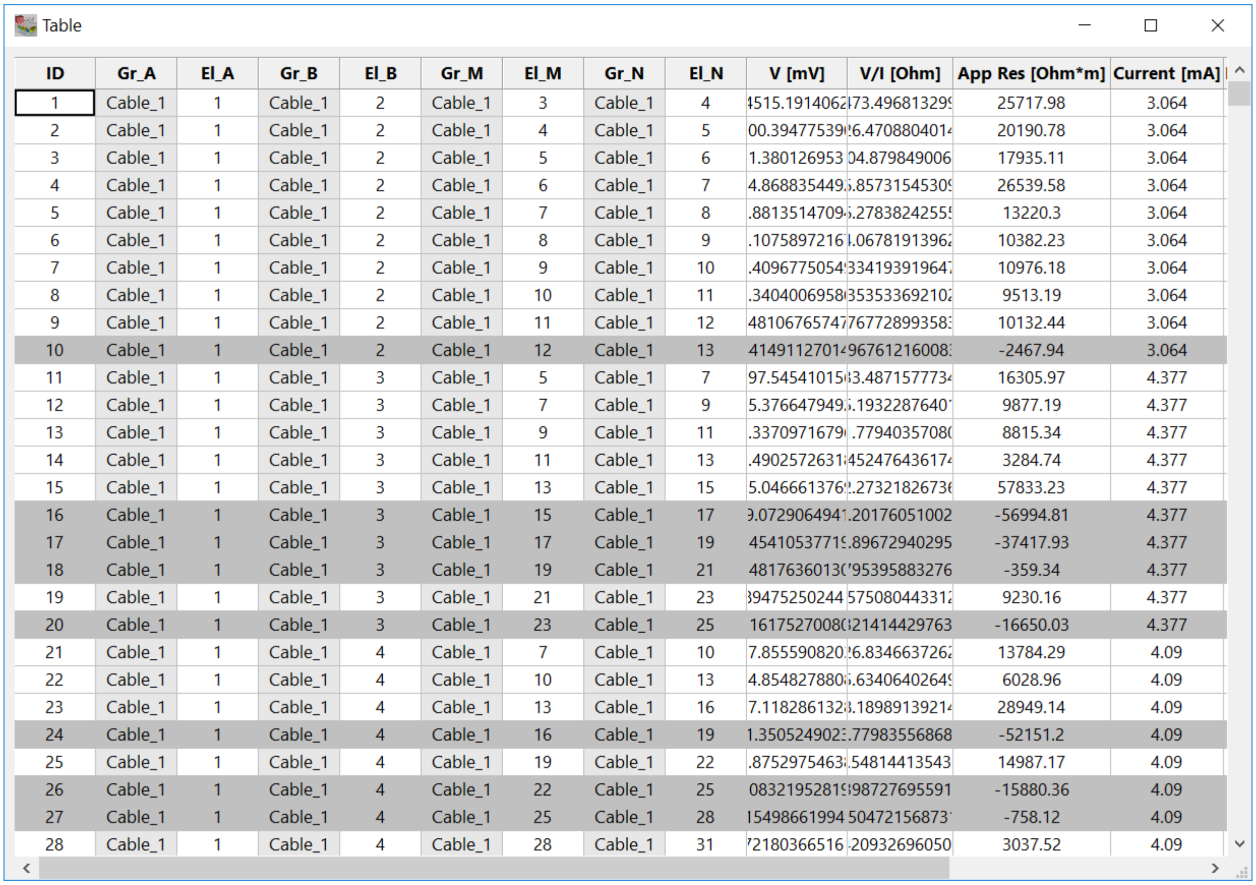
Note that this values can be also graphically set with the use of the mouse (and its left button).

19.3.5 Apply

After that the filtering parameters are set, then it is possible to *Apply* the process pressing this button.

An estimation of the number of measurement that will be removed it is written at the right of the button.

The filtered data are not deleted only excluded from the inversion processing. The table is automatically updated and the filtered data are greyed out (Figure 19.16).



Table

ID	Gr_A	El_A	Gr_B	El_B	Gr_M	El_M	Gr_N	El_N	V [mV]	V/I [Ohm]	App Res [Ohm*m]	Current [mA]
1	Cable_1	1	Cable_1	2	Cable_1	3	Cable_1	4	4515.191406	73.49681329	25717.98	3.064
2	Cable_1	1	Cable_1	2	Cable_1	4	Cable_1	5	00.39477539	6.470880401	20190.78	3.064
3	Cable_1	1	Cable_1	2	Cable_1	5	Cable_1	6	1.380126953	04.879849006	17935.11	3.064
4	Cable_1	1	Cable_1	2	Cable_1	6	Cable_1	7	4.868835449	.8573154530	26539.58	3.064
5	Cable_1	1	Cable_1	2	Cable_1	7	Cable_1	8	.8813514709	.2783824255	13220.3	3.064
6	Cable_1	1	Cable_1	2	Cable_1	8	Cable_1	9	.1075897216	.0678191396	10382.23	3.064
7	Cable_1	1	Cable_1	2	Cable_1	9	Cable_1	10	.4096775054	33419391964	10976.18	3.064
8	Cable_1	1	Cable_1	2	Cable_1	10	Cable_1	11	.3404006958	35353369210	9513.19	3.064
9	Cable_1	1	Cable_1	2	Cable_1	11	Cable_1	12	4810676574	76772899358	10132.44	3.064
10	Cable_1	1	Cable_1	2	Cable_1	12	Cable_1	13	4149112701	96761216008	-2467.94	3.064
11	Cable_1	1	Cable_1	3	Cable_1	5	Cable_1	7	97.54541015	3.487157773	16305.97	4.377
12	Cable_1	1	Cable_1	3	Cable_1	7	Cable_1	9	5.376647949	.1932287640	9877.19	4.377
13	Cable_1	1	Cable_1	3	Cable_1	9	Cable_1	11	.3370971679	.7794035708	8815.34	4.377
14	Cable_1	1	Cable_1	3	Cable_1	11	Cable_1	13	.4902572631	45247643617	3284.74	4.377
15	Cable_1	1	Cable_1	3	Cable_1	13	Cable_1	15	5.046661376	.2732182673	57833.23	4.377
16	Cable_1	1	Cable_1	3	Cable_1	15	Cable_1	17	9.072906494	.20176051002	-56994.81	4.377
17	Cable_1	1	Cable_1	3	Cable_1	17	Cable_1	19	4541053771	.89672940295	-37417.93	4.377
18	Cable_1	1	Cable_1	3	Cable_1	19	Cable_1	21	4817636013	95395883276	-359.34	4.377
19	Cable_1	1	Cable_1	3	Cable_1	21	Cable_1	23	39475250244	57508044331	9230.16	4.377
20	Cable_1	1	Cable_1	3	Cable_1	23	Cable_1	25	1617527008	21414429763	-16650.03	4.377
21	Cable_1	1	Cable_1	4	Cable_1	7	Cable_1	10	7.855590820	6.834663726	13784.29	4.09
22	Cable_1	1	Cable_1	4	Cable_1	10	Cable_1	13	4.854827880	.6340640264	6028.96	4.09
23	Cable_1	1	Cable_1	4	Cable_1	13	Cable_1	16	7.118286132	.1898913921	28949.14	4.09
24	Cable_1	1	Cable_1	4	Cable_1	16	Cable_1	19	1.350524902	.77983556868	-52151.2	4.09
25	Cable_1	1	Cable_1	4	Cable_1	19	Cable_1	22	.8752975463	.54814413543	14987.17	4.09
26	Cable_1	1	Cable_1	4	Cable_1	22	Cable_1	25	0832195281	98727695591	-15880.36	4.09
27	Cable_1	1	Cable_1	4	Cable_1	25	Cable_1	28	15498661994	50472156873	-758.12	4.09
28	Cable_1	1	Cable_1	4	Cable_1	28	Cable_1	31	72180366516	20932696050	3037.52	4.09

Figure 19.16: In grey, filtered measurement

However, it is possible to delete them from the project (see section *Delete*).

19.3.6 Preset

If some filter is commonly used, or a sequence of them is frequent, then it can be saved and recalled with a quicker procedure.

First it is necessary to set the parameter of of the filter (see *Measurement Type*, *Mode* and *Min Max*). After this in place of press the button *Apply* press the button "*Save/Append*". Because more *Preset* can be specified then the name needs to be specified.

After the the filter(s) are specified it is then later possible to execute it pressing the button "*Execute...*"; it will be asked also the the name of the preset to be run.

Pressing the button "*View...*" it is possible to examine an already saved *Preset*, to possibly modify it also.

19.3.7 Unskip All

In case of problem it is possible to reset all the filtering steps and so to restore all the measurement in the dataset.

19.3.8 Capture

Pressing this button it is possible to save to file the current plot shown.

19.4 Scatter Plot

This option visualizes the measurements as X-Y dispersion plot, to have a graphical representation of data distribution.

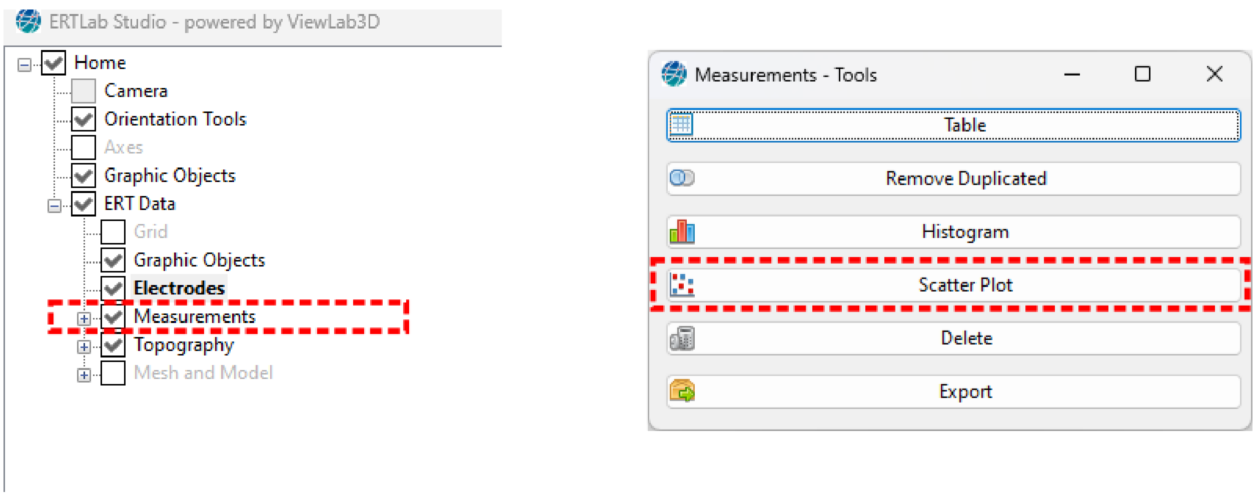


Figure 19.17: Scatter plot Panel

In this window two measurements can be visualized as scatterplot.

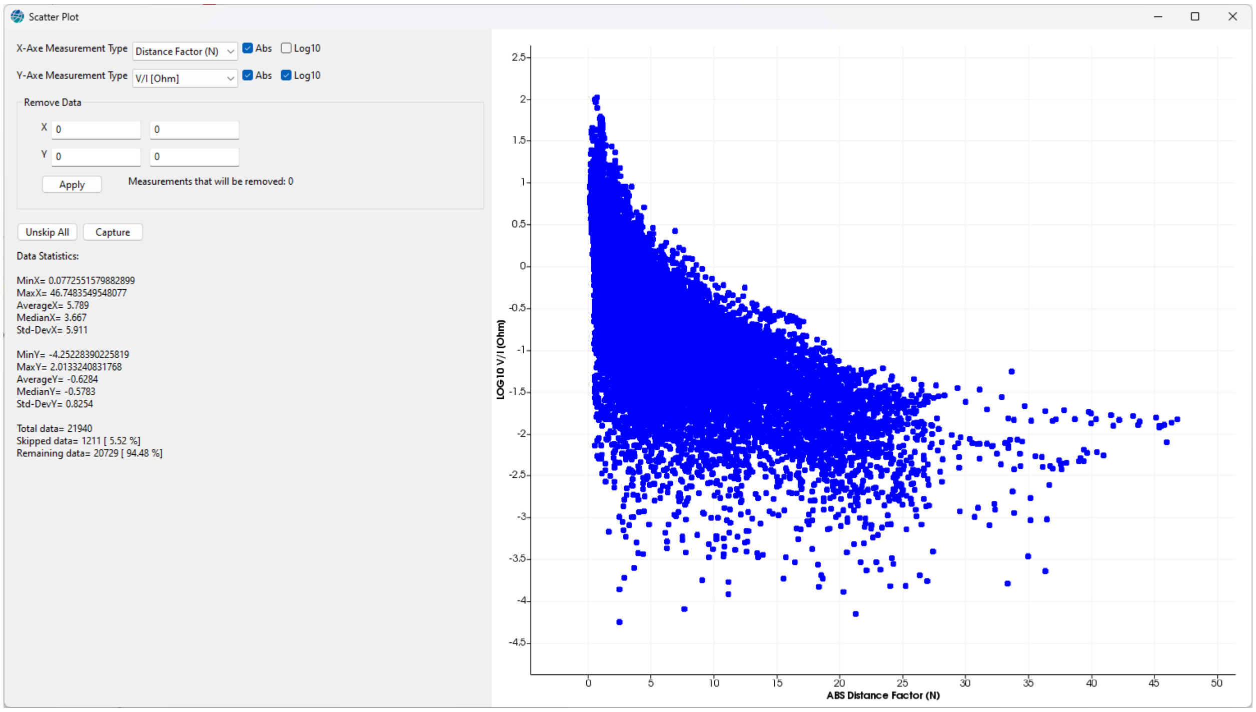


Figure 19.18: Scatter plot

On the left side the two types of measurements to display can be selected, one for the X-axis and one for the Y-axis. Click in the dropdown boxes for the possible selections, which are:

- V/I [Ohm]
- Apparent Resistivity [Ohm*m]
- IP [mV/V]
- K [m]
- V [mV]
- I [mA]
- Standard Deviation V/I [Ohm]
- Standard Deviation V[V]
- Standard Deviation IP [mV/V]
- Standard Deviation IP [%]
- Calculated V/I [Ohm]
- Calculated Apparent Resistivity [Ohm*m]
- Calculated IP [mV/V]
- Calculated Standard Deviation V/I [Ohm]

- Calculated Standard Deviation IP [mV/V]
- Calculated Standard Deviation IP [%]
- ID
- EI A ID
- EI B ID
- EI N ID
- X
- Y
- Z

19.4.1 Remove Data

Here can be set a range of values (both in terms of X and Y axes) where the filter will remove the inner measurements.

Note that this values can be also graphically set with the use of the mouse (and its left button).

19.4.2 Apply

After that the filtering parameters are set, then it is possible to *Apply* the process pressing this button.

An estimation of the number of measurement that will be removed it is written at the right of the button.

19.4.3 Unskip All

In case of problem it is possible to reset all the filtering steps and so to restore all the measurement in the dataset.

19.4.4 Capture

Pressing this button it is possible to save to file the current plot shown.

19.5 Delete

This option allows to delete some measurements. Click on the *Delete* button; an information window will show the total number of measurements and the number of filtered measurements.

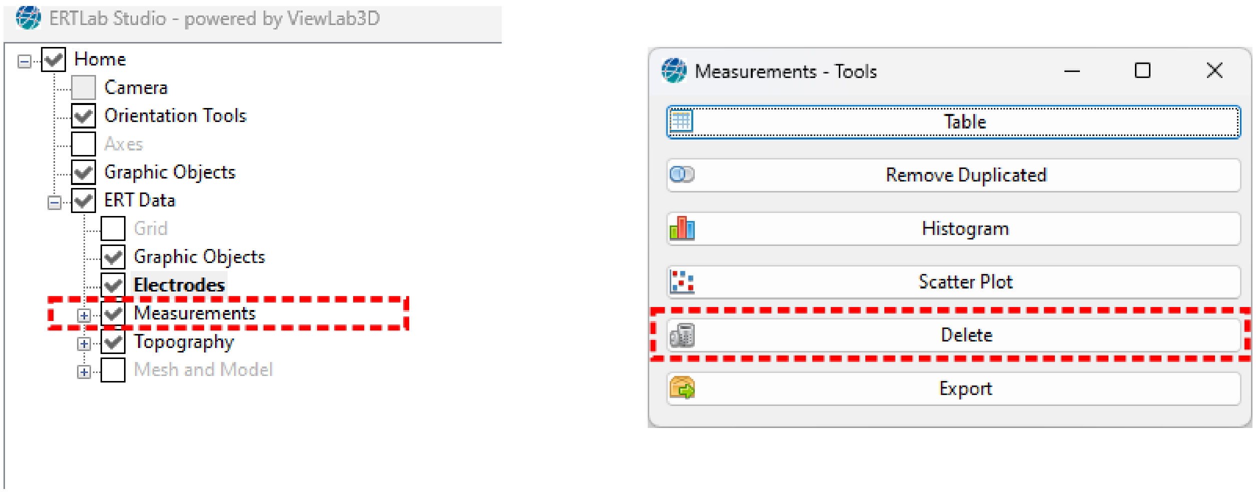


Figure 19.19: Delete measurements button

Note, if there are no filtered data, clicking on OK button will delete all the data (Figure 19.20).

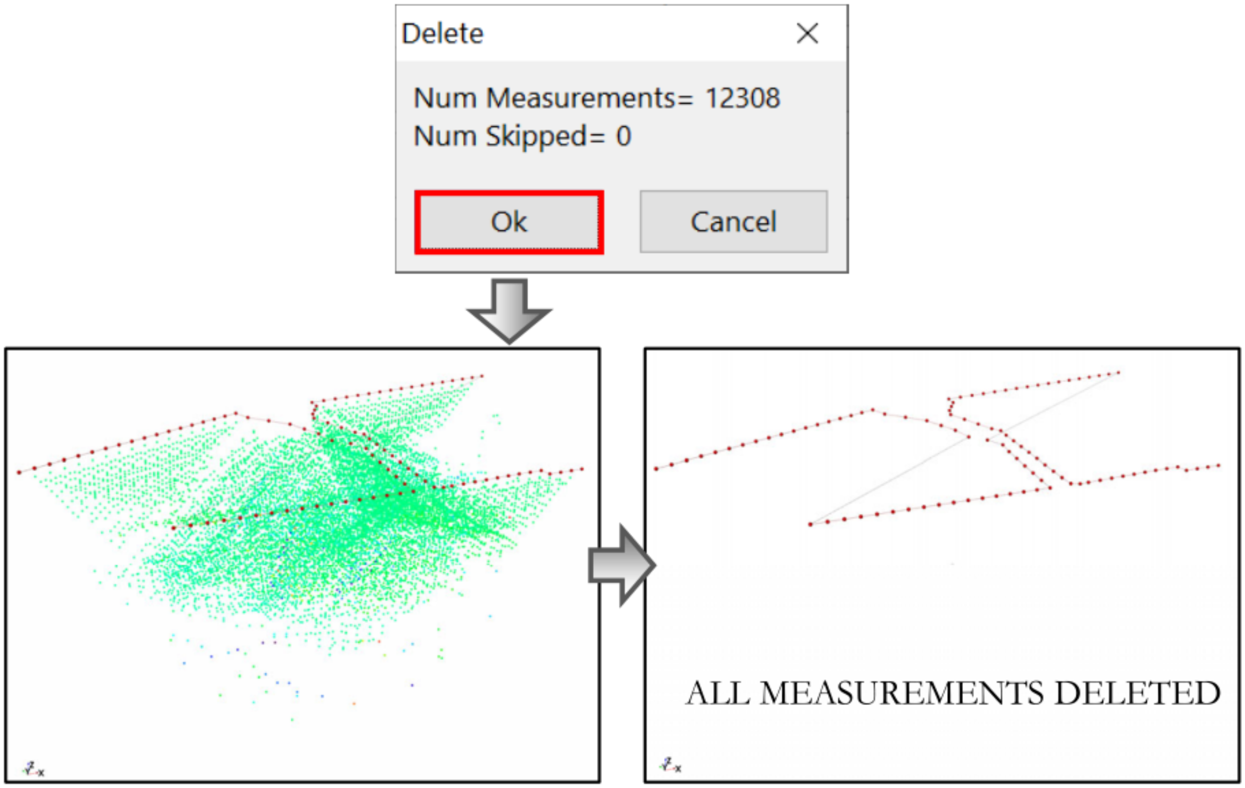


Figure 19.20: Delete measurement with no skipped data

On the contrary, if some filtered data are present one can choose to delete them or those remaining (Figure 19.21). Clicking on the OK button deletes the measurements from the dataset.

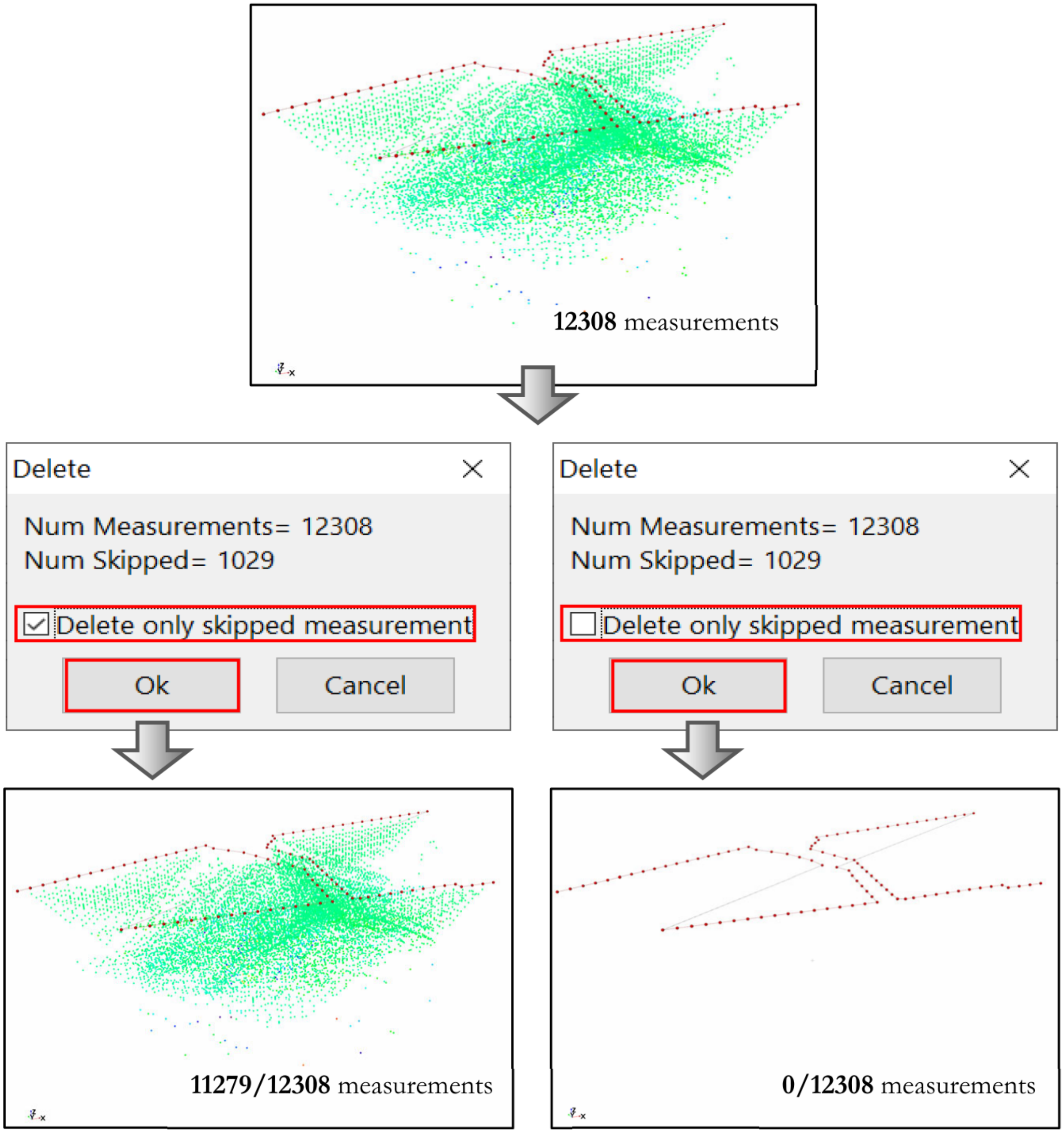


Figure 19.21: Remove measurements with skipped data

19.6 Export

Through this option it is possible to export a .txt file with information about electrodes and measurements.

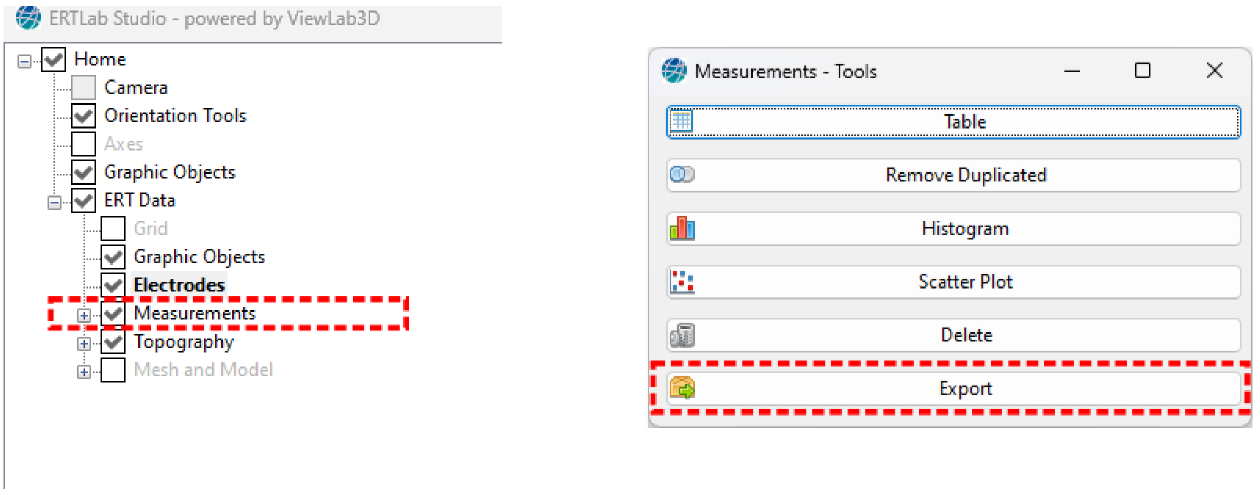
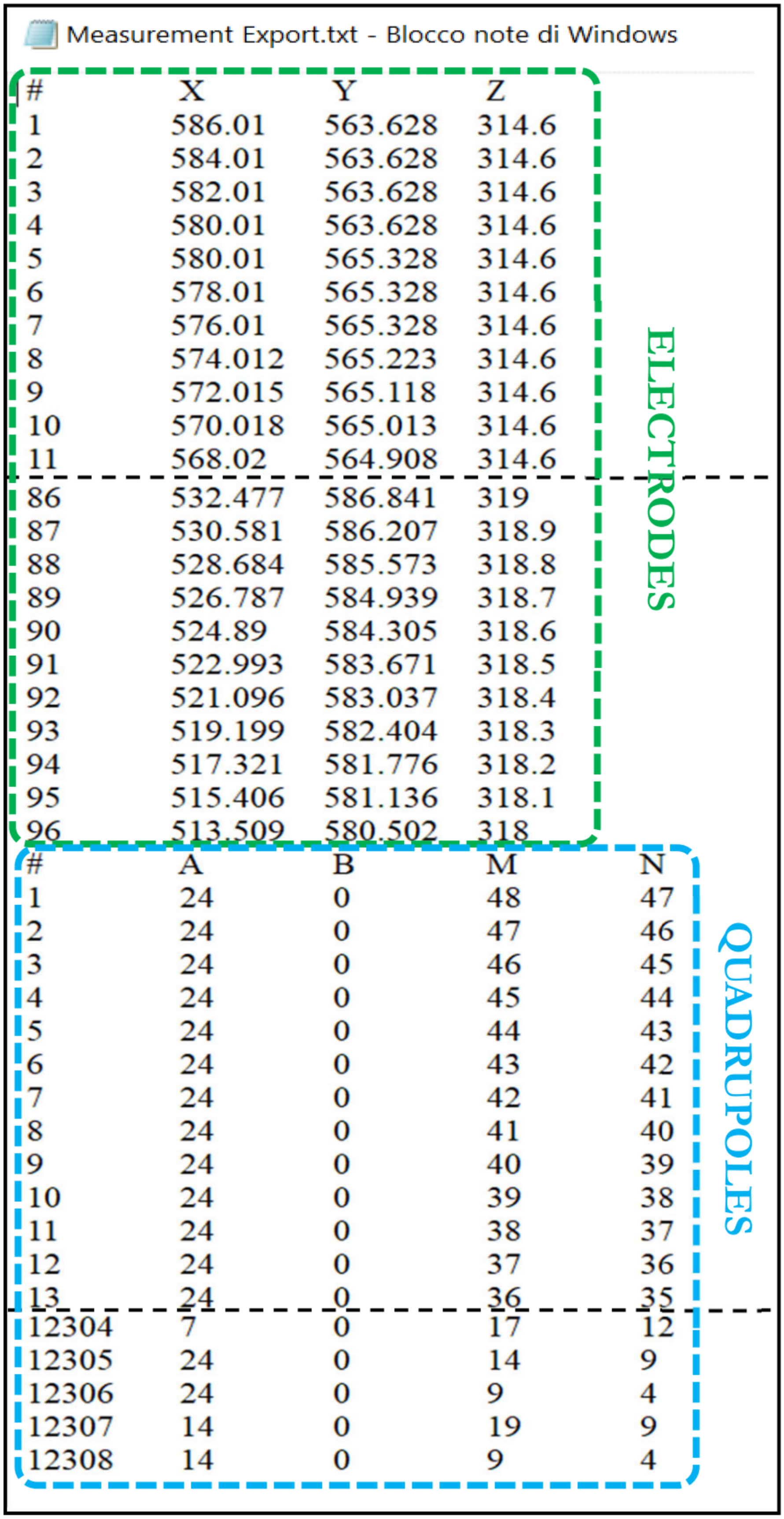


Figure 19.22: Export button

The saved file is a .txt file where the first part shows the coordinates of the electrodes and in the next part the electrodes that constitute the quadrupole ABMN of each measurement (Figure 19.23).



Measurement Export.txt - Blocco note di Windows

#	X	Y	Z
1	586.01	563.628	314.6
2	584.01	563.628	314.6
3	582.01	563.628	314.6
4	580.01	563.628	314.6
5	580.01	565.328	314.6
6	578.01	565.328	314.6
7	576.01	565.328	314.6
8	574.012	565.223	314.6
9	572.015	565.118	314.6
10	570.018	565.013	314.6
11	568.02	564.908	314.6
86	532.477	586.841	319
87	530.581	586.207	318.9
88	528.684	585.573	318.8
89	526.787	584.939	318.7
90	524.89	584.305	318.6
91	522.993	583.671	318.5
92	521.096	583.037	318.4
93	519.199	582.404	318.3
94	517.321	581.776	318.2
95	515.406	581.136	318.1
96	513.509	580.502	318

#	A	B	M	N
1	24	0	48	47
2	24	0	47	46
3	24	0	46	45
4	24	0	45	44
5	24	0	44	43
6	24	0	43	42
7	24	0	42	41
8	24	0	41	40
9	24	0	40	39
10	24	0	39	38
11	24	0	38	37
12	24	0	37	36
13	24	0	36	35
12304	7	0	17	12
12305	24	0	14	9
12306	24	0	9	4
12307	14	0	19	9
12308	14	0	9	4

Figure 19.23: Export measurements table

CHAPTER TWENTY

TOPOGRAPHY TOOLS

Right clicking on the node the "*Topography*" option, a window will appear with the available tools (Figure 20.1).

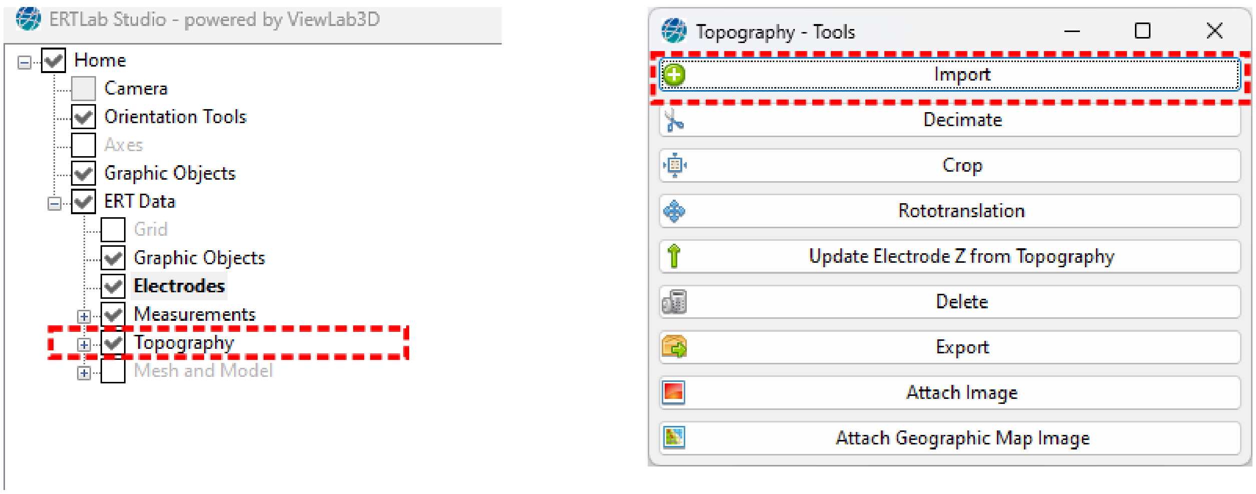
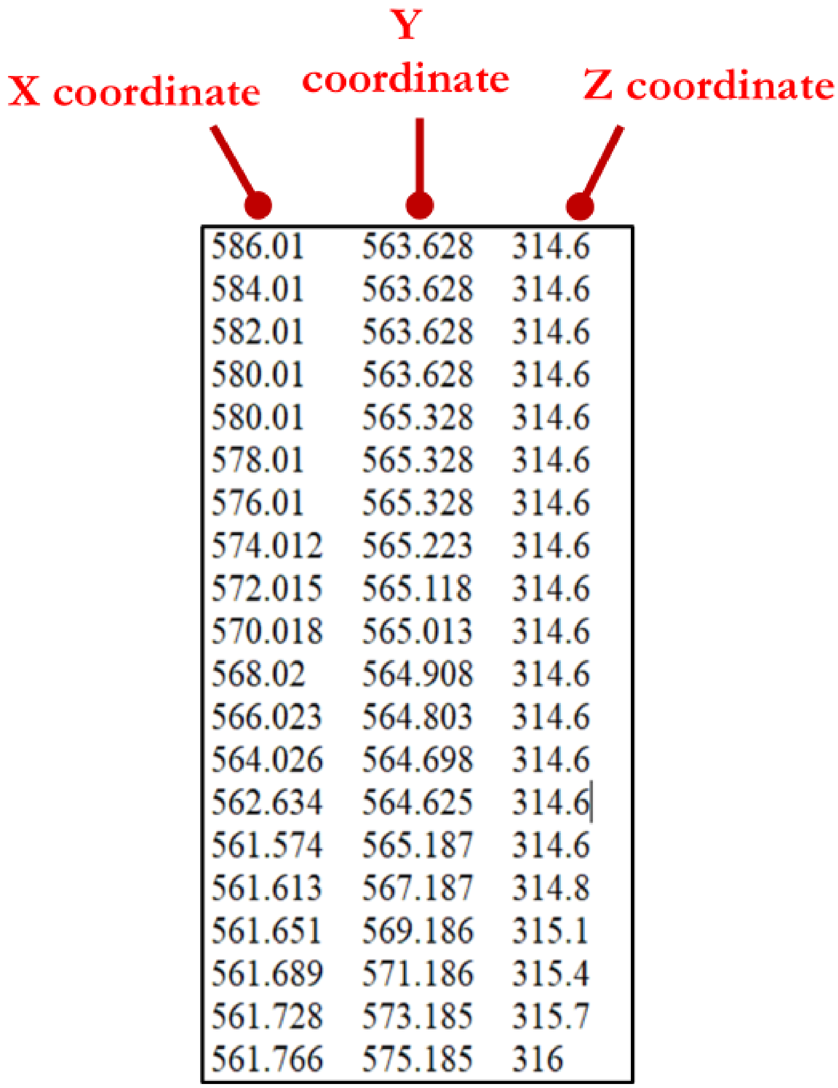


Figure 20.1: Topography tools panel

20.1 Import

To import a topography, load a .txt file with 3 columns, one for each coordinate (X, Y, Z) (Figure 20.2). Each row of the topography file corresponds to one topographic point.



586.01	563.628	314.6
584.01	563.628	314.6
582.01	563.628	314.6
580.01	563.628	314.6
580.01	565.328	314.6
578.01	565.328	314.6
576.01	565.328	314.6
574.012	565.223	314.6
572.015	565.118	314.6
570.018	565.013	314.6
568.02	564.908	314.6
566.023	564.803	314.6
564.026	564.698	314.6
562.634	564.625	314.6
561.574	565.187	314.6
561.613	567.187	314.8
561.651	569.186	315.1
561.689	571.186	315.4
561.728	573.185	315.7
561.766	575.185	316

Figure 20.2: Example of Topographic file

If the loaded file is composed by a high number of points, with excessive spatial definition not needed for the ERT work purpose, ERTLab *Studio* automatically suggests a decimation of it (Figure 20.3).

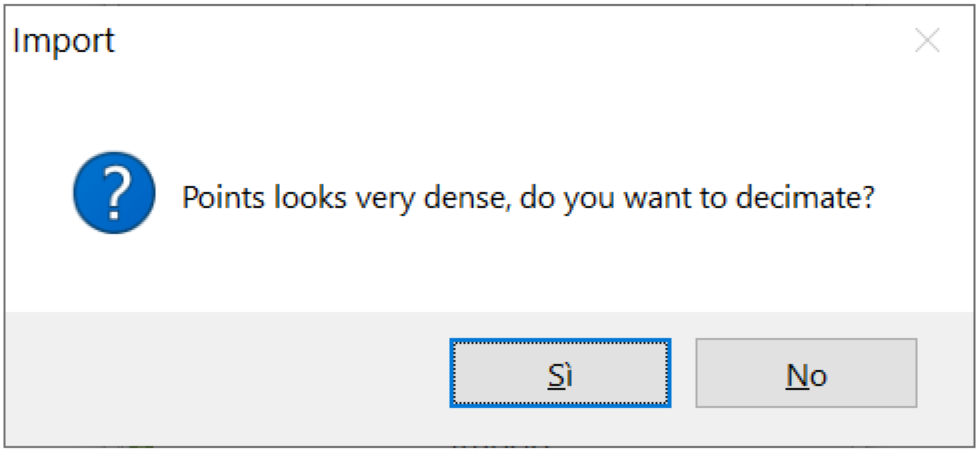


Figure 20.3: Automatic decimation of Topography file

It is also possible to do this later through the dedicated tool (see *Decimate*).

20.2 Decimate

If the topography file is too big and slowing-down the application, the file can be decimated through this option.

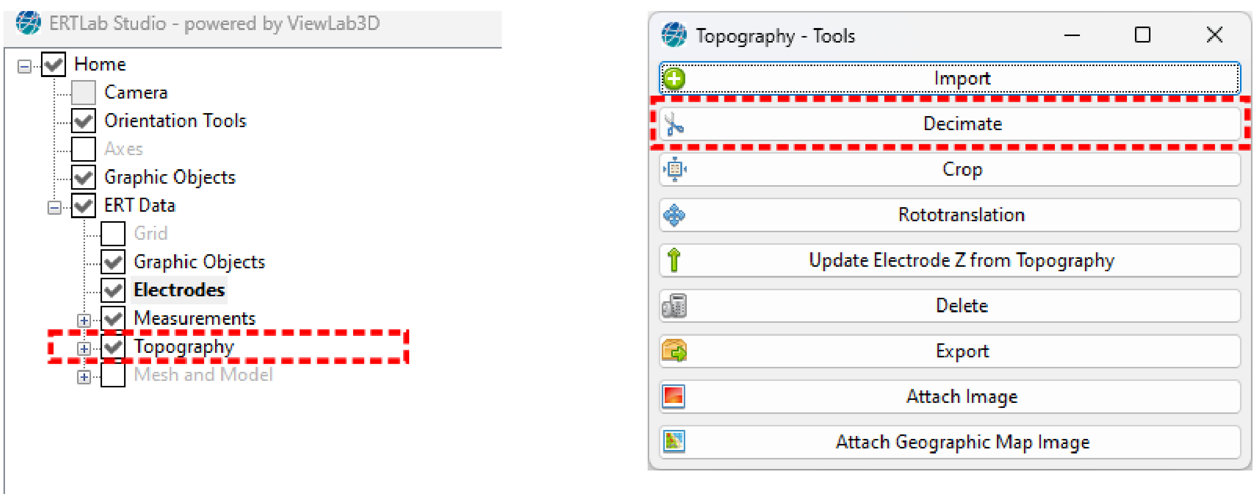


Figure 20.4: Topography tools panel

Clicking on the *Decimate* button will open a window (Figure 20.5).

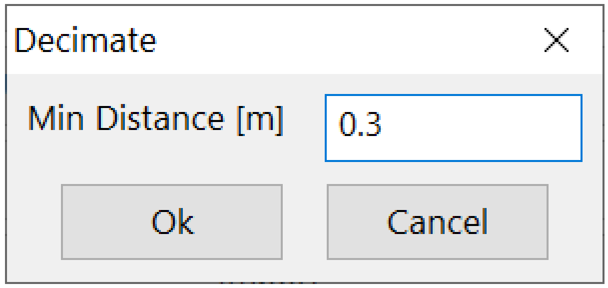


Figure 20.5: Decimate radius

Topographic points located at a distance smaller than indicated in the *Min Distance* box are deleted. Be careful not too delete to many points to not decimate the file too much to not lose the topography resolution.

20.3 Crop

If the topography data points extent too much beyond the ERT area of interest the topography can be cropped with this option.

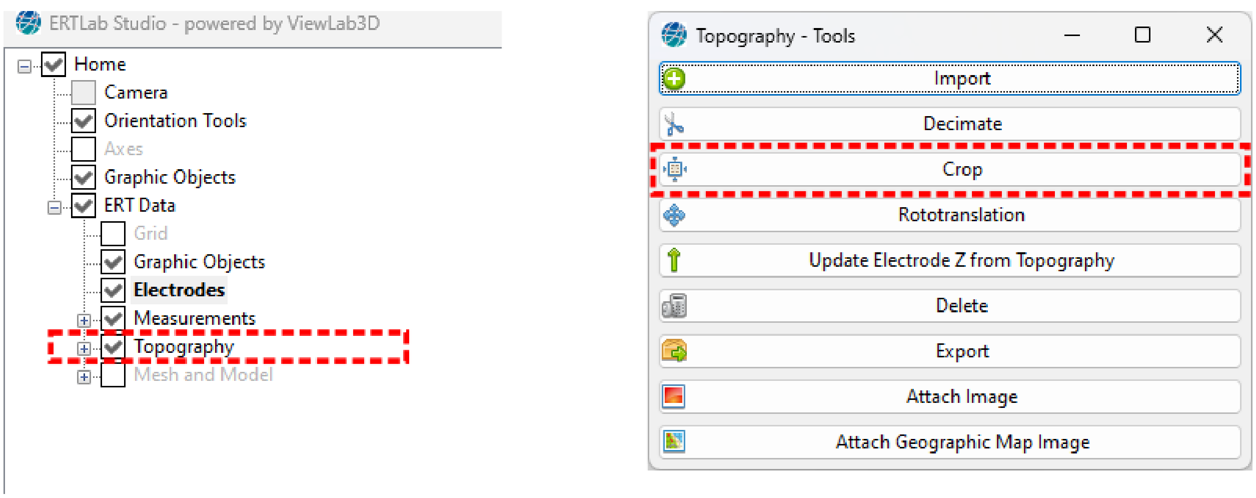


Figure 20.6: Crop tool

Figure 20.7 shows a big topography surface with an ERT line in loop configuration in a small area.

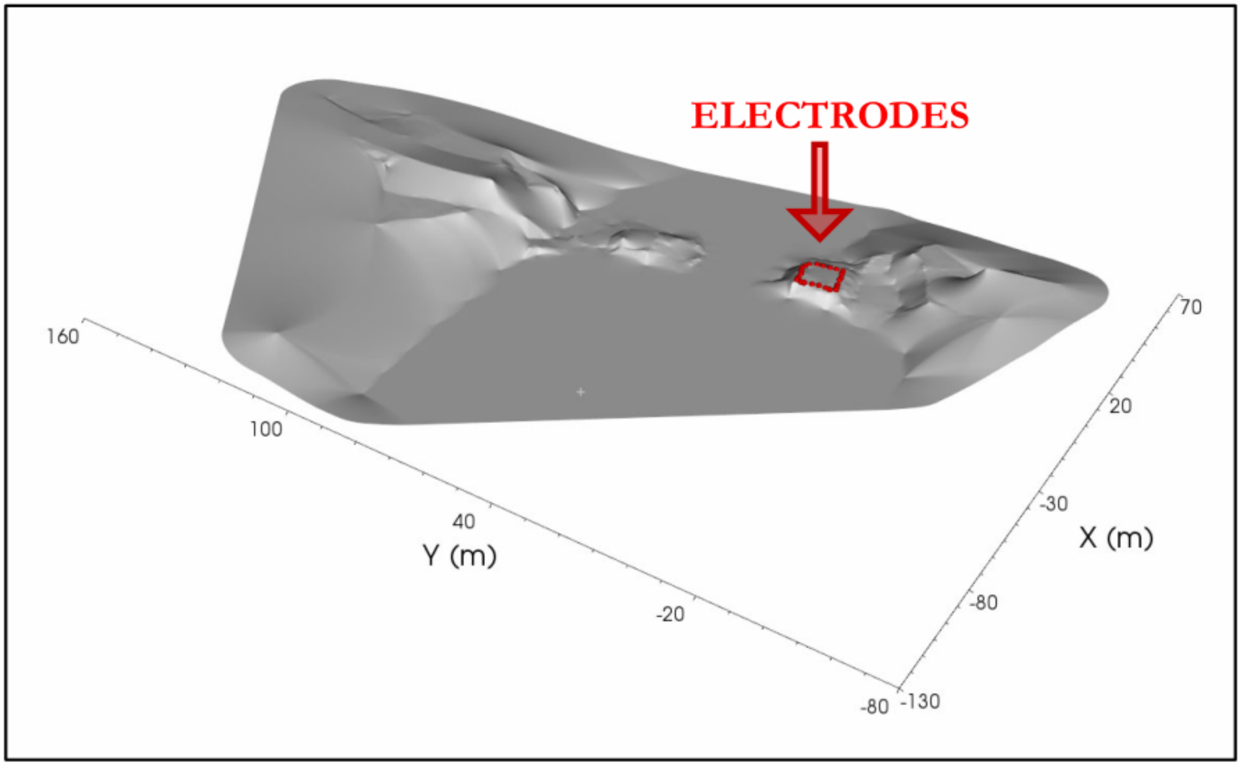


Figure 20.7: Example of extended topography

Using the *Crop* button allows to cut it typing the proper coordinates in the number boxes (Figure 20.9).

To find the maximum and minimum crop values use the *Picker tool* (see section View Picker settings).

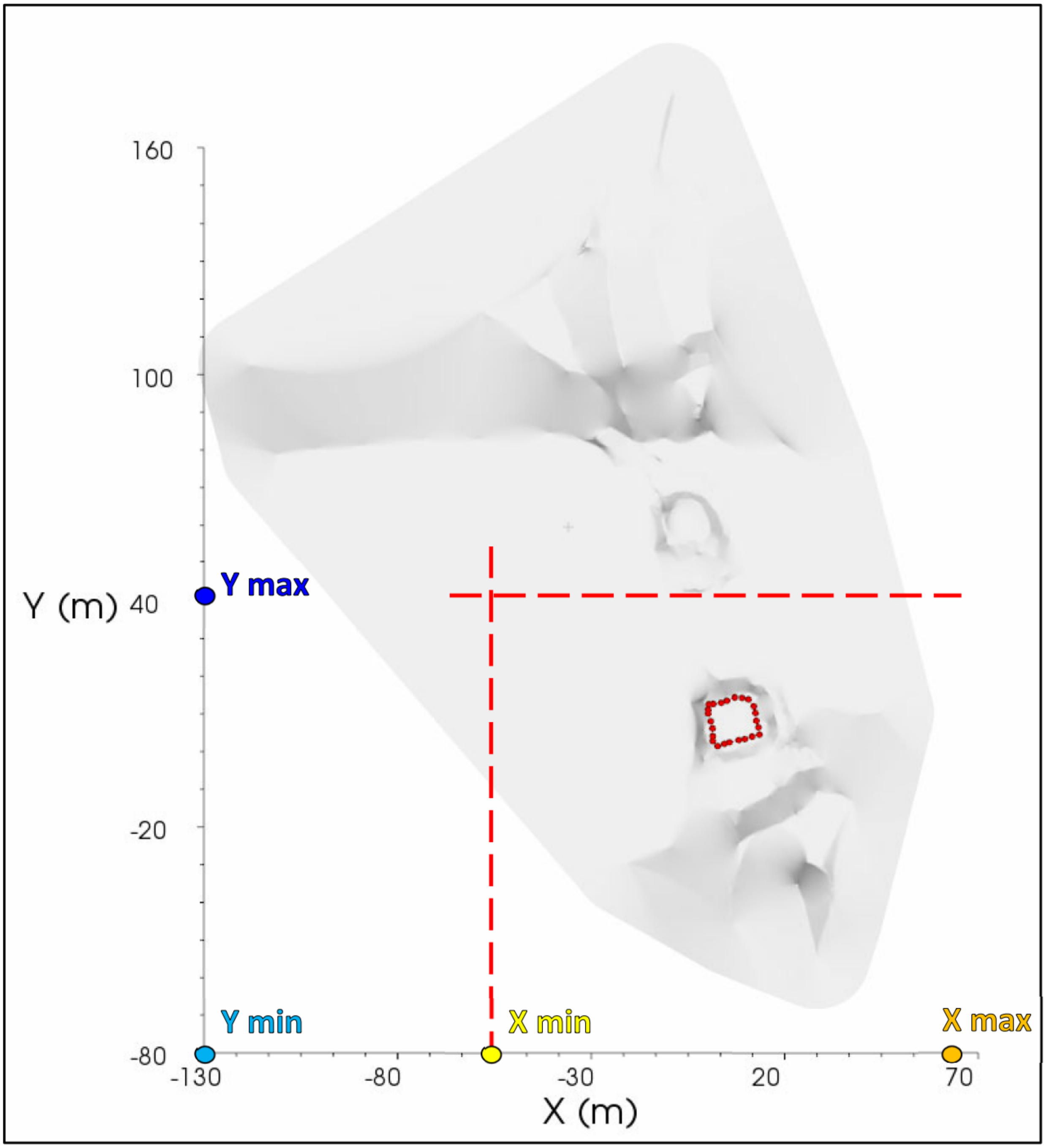


Figure 20.8: Choice of cut values to focus on the area of interest

Checking the *keep inside* box (Figure 20.9), all the points of the topography which are outside the selected limits are deleted.

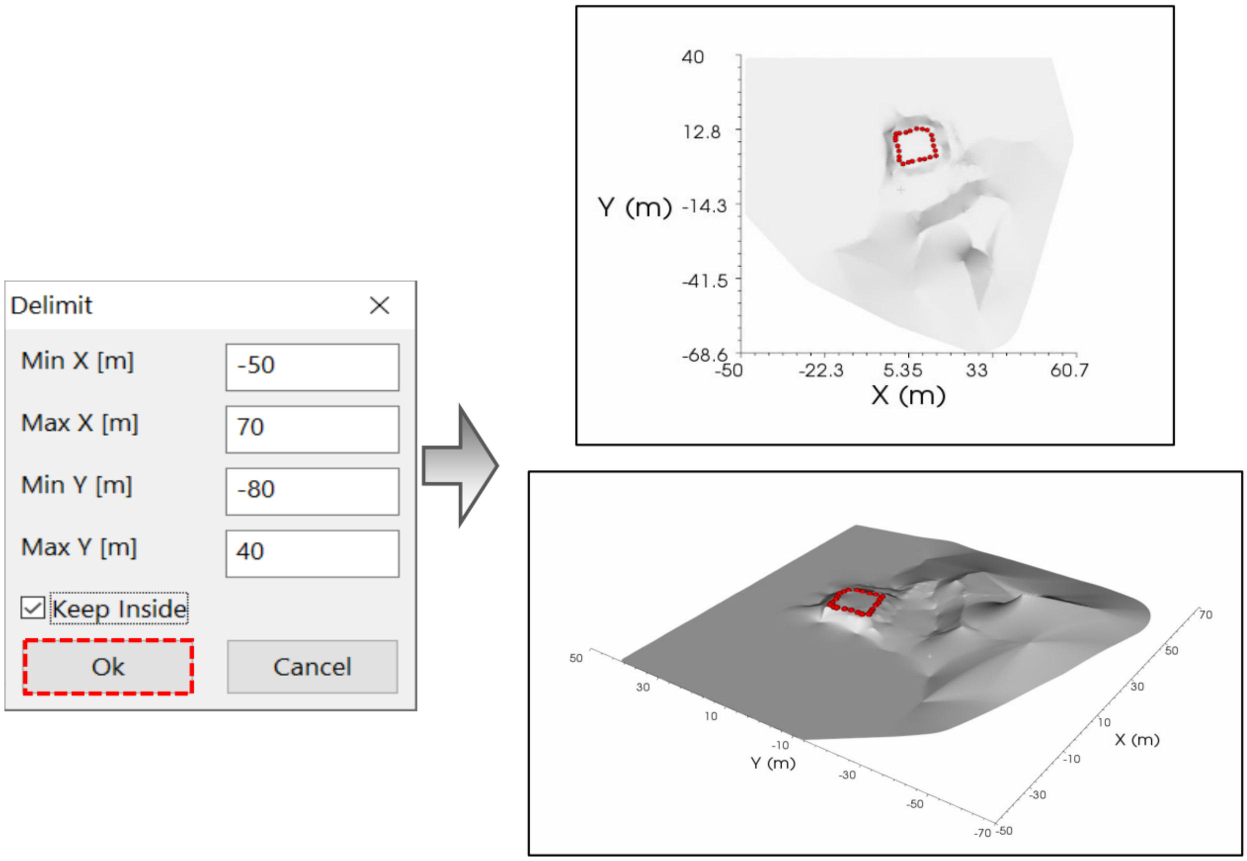


Figure 20.9: Result of topography crop

20.4 Rototranslation

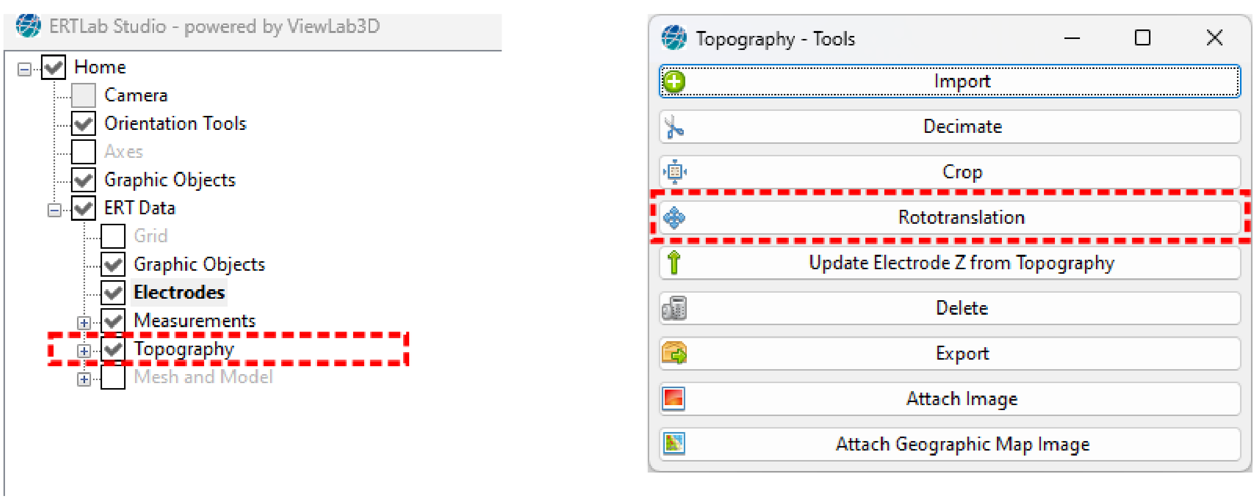


Figure 20.10: Rototranslation tool

This option allows the translation and rotation of the Topography in the 3D dimensions (see section Rototranslation). It performs a real rototranslation and not only a graphic one, so when exporting the coordinates in a .txt files after having applied a rototranslation they are different from those imported.

20.5 Update Electrodes Z from topography

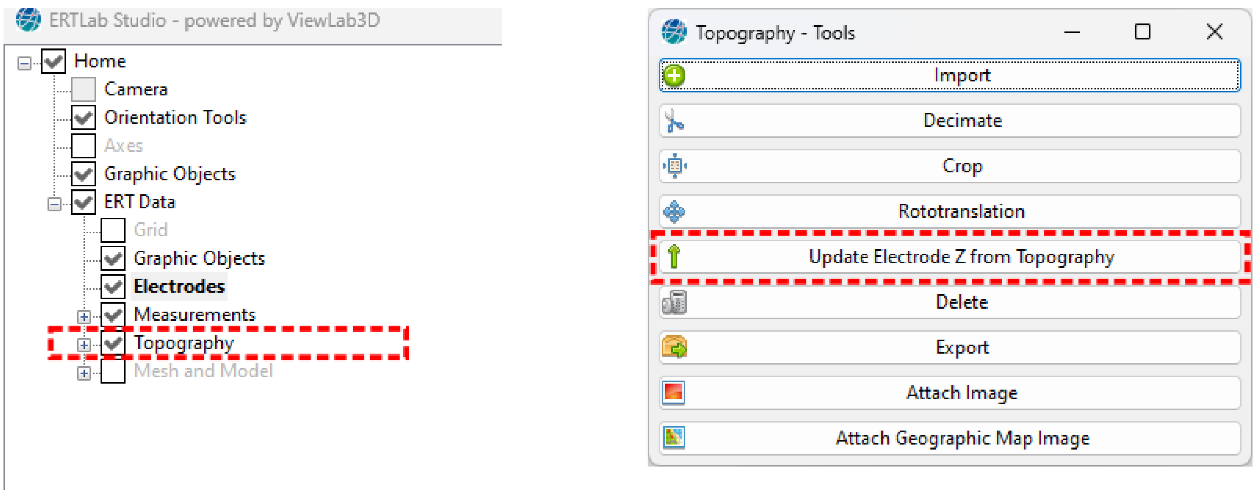


Figure 20.11: Update Z tool

When a Topography is loaded, it can happen that one or more electrodes are not coincident with the Z coordinate of the topography. They can appear elevated or buried. With this option the Z of the electrodes can be updated to match the topography. In the following example, some elevated electrodes are on the topographic Z surface after the application of this option.

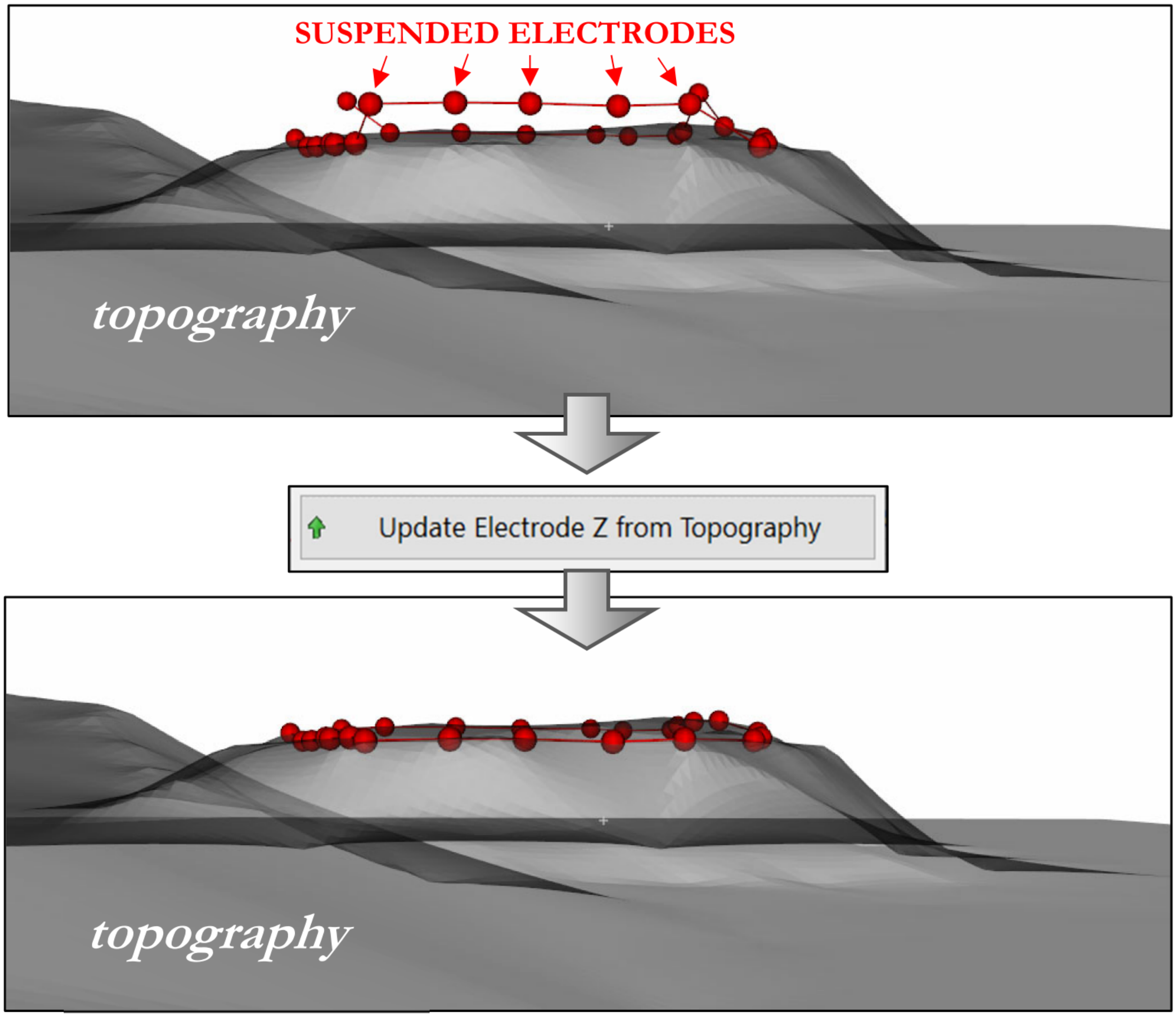


Figure 20.12: Example of Z electrode update from Topography

20.6 Delete

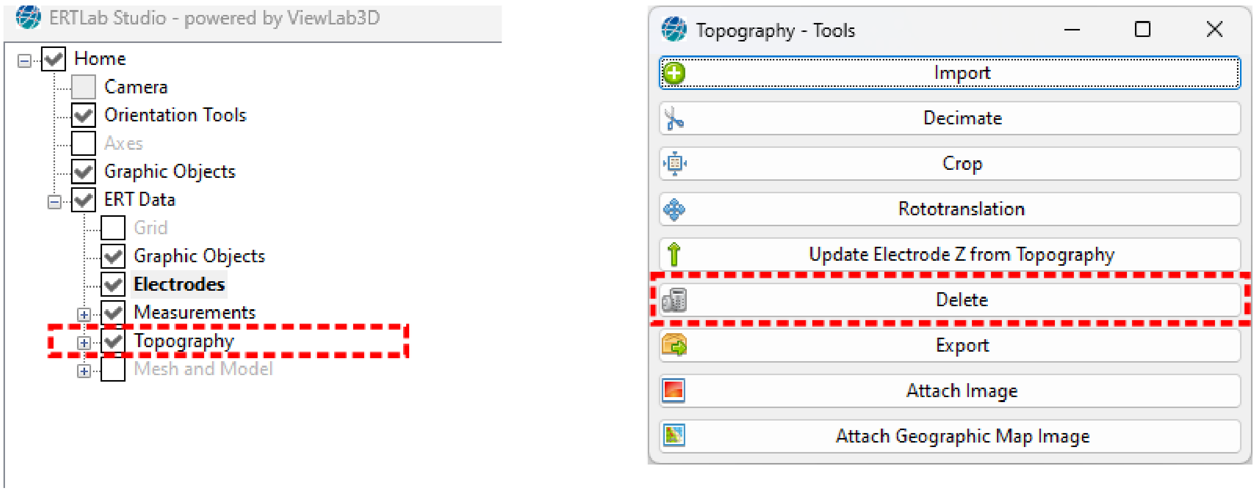


Figure 20.13: Delete tool

Clicking on the *Delete* button deletes all the points of the topography and there is no longer any topographic information.

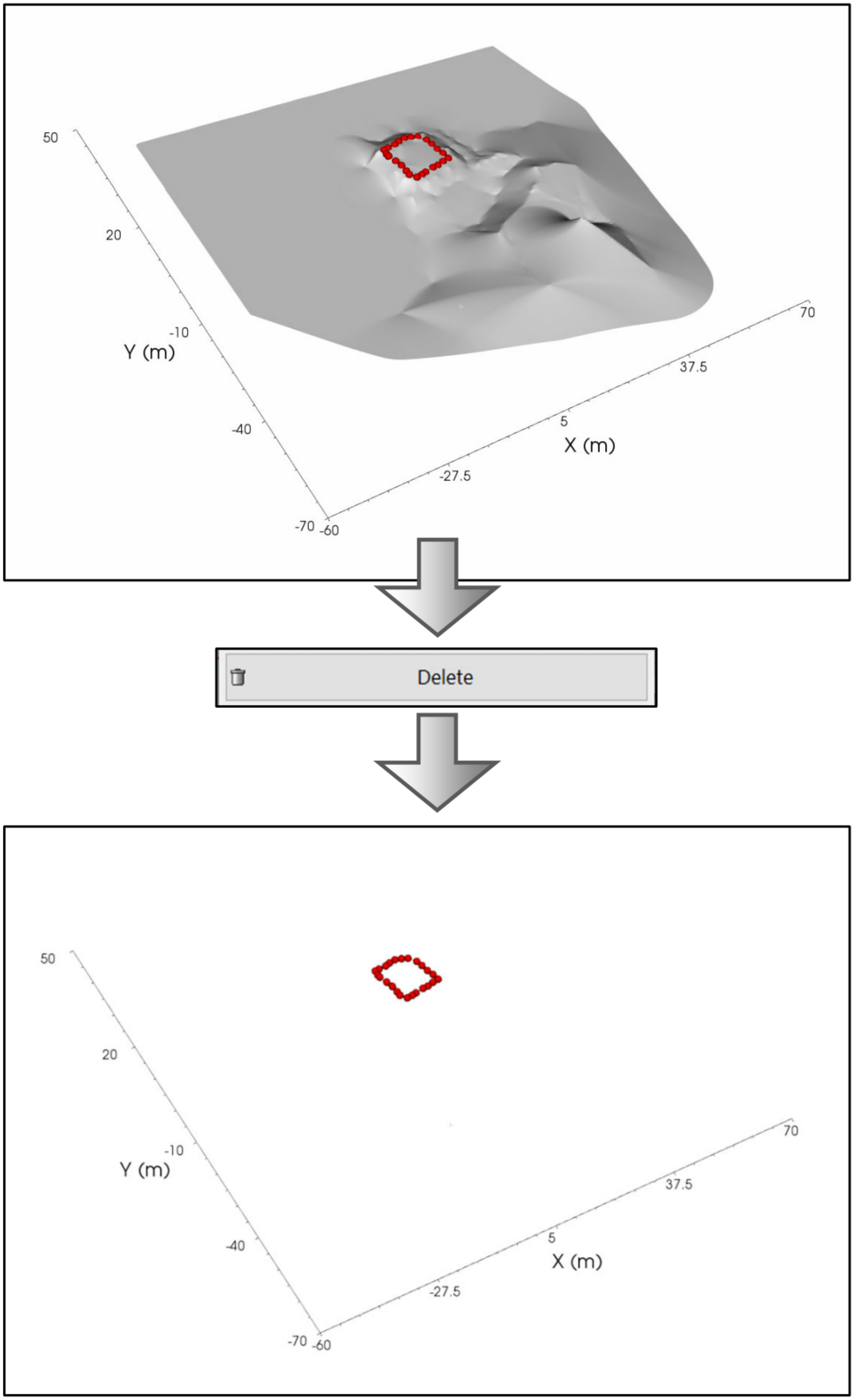


Figure 20.14: Example of delete Topography

20.7 Export

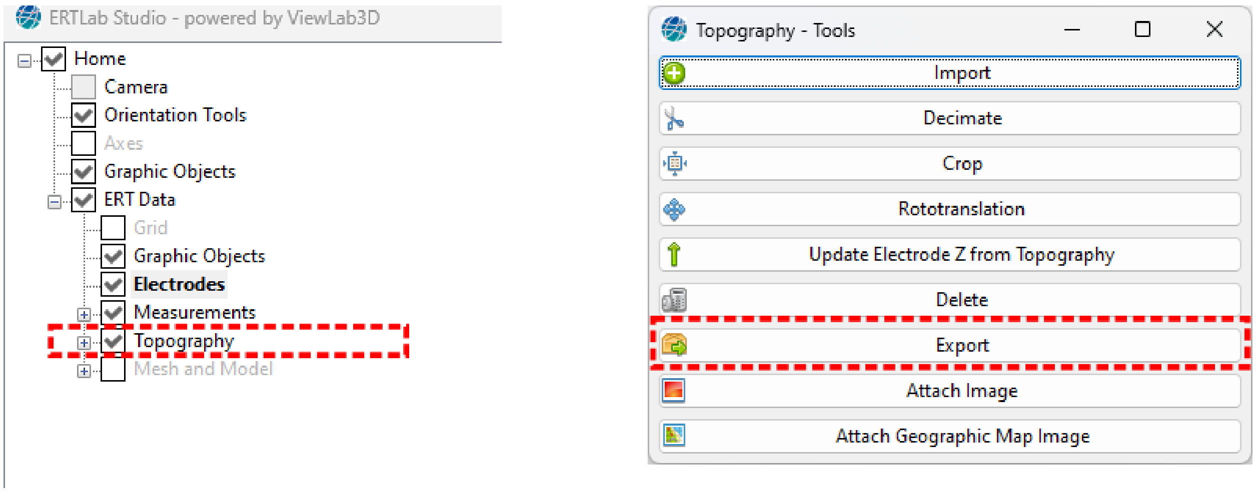


Figure 20.15: Export tool

Clicking on the *Export* button all the points of the topography can be saved to a text file.

20.8 Attach Image

Warning:	Feature not enabled in the default license type

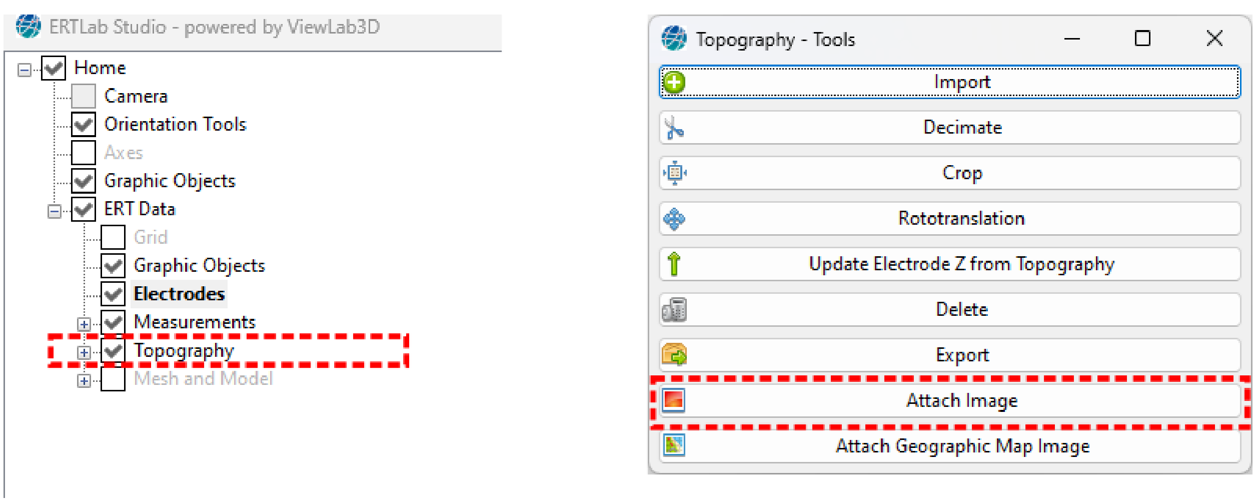


Figure 20.16: Image tool

By default the topography surface has an homogeneous colour, in some cases can be useful to paint that surface with some image. After that the image is selected a new child node in the tree is created, there the image can be customized (see Image).

Note that by default the size of the image is 1m by 1m, placed at the centre of the scene, so it is often needed to edit it.

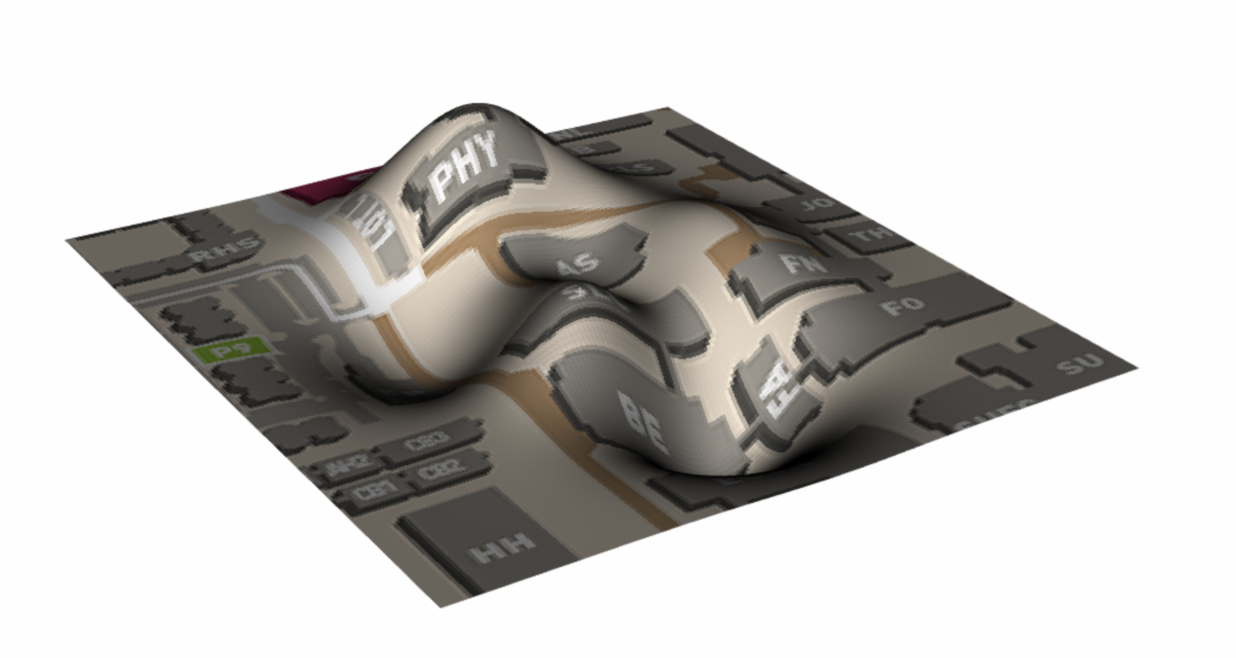


Figure 20.17: Example of Image on Topography

20.9 Attach Geographic Map Image

Warning: Feature not enabled in the default license type

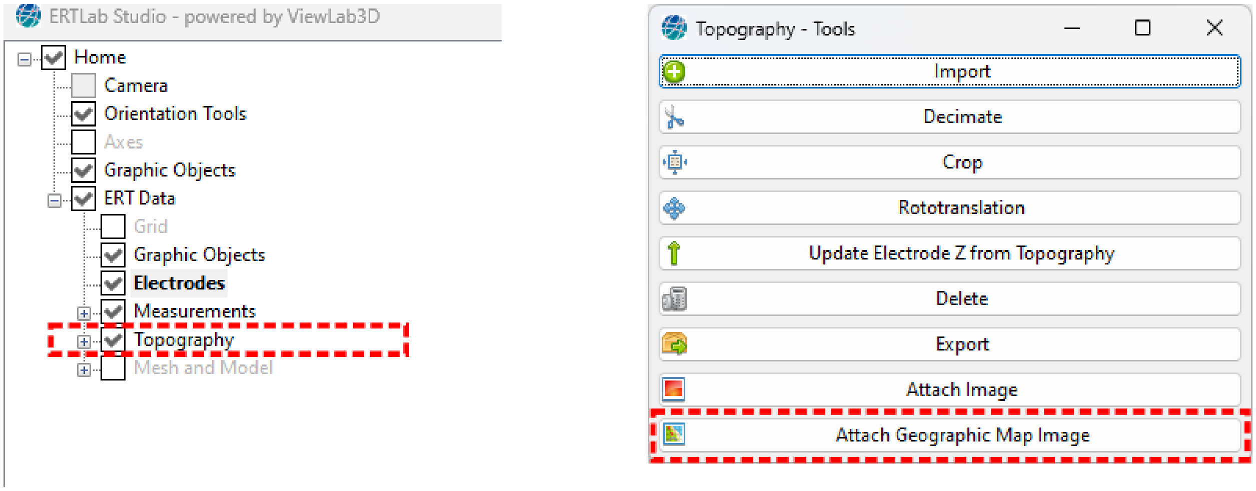


Figure 20.18: Geographic Map tool

By default the topography surface has an homogeneous colour, in some cases can be useful to paint that surface with some map. A new child node in the tree is created, there the map can be customized (see Geographic Map Image).

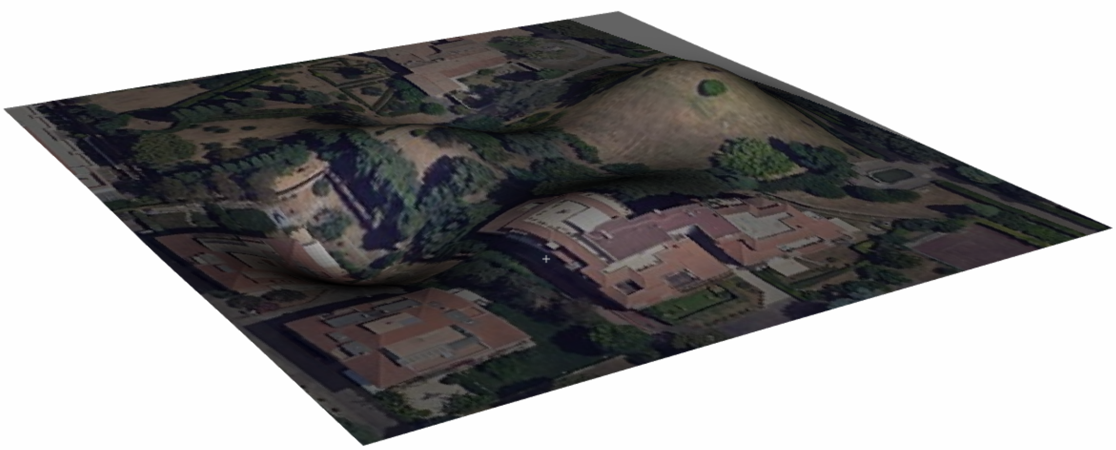


Figure 20.19: Example of Geographic Image on Topography

CHAPTER

TWENTYONE

MESH AND MODEL TOOLS

Right clicking on the node the “*Mesh and Model*” option, a window will appear with the available tools (Figure 21.1).

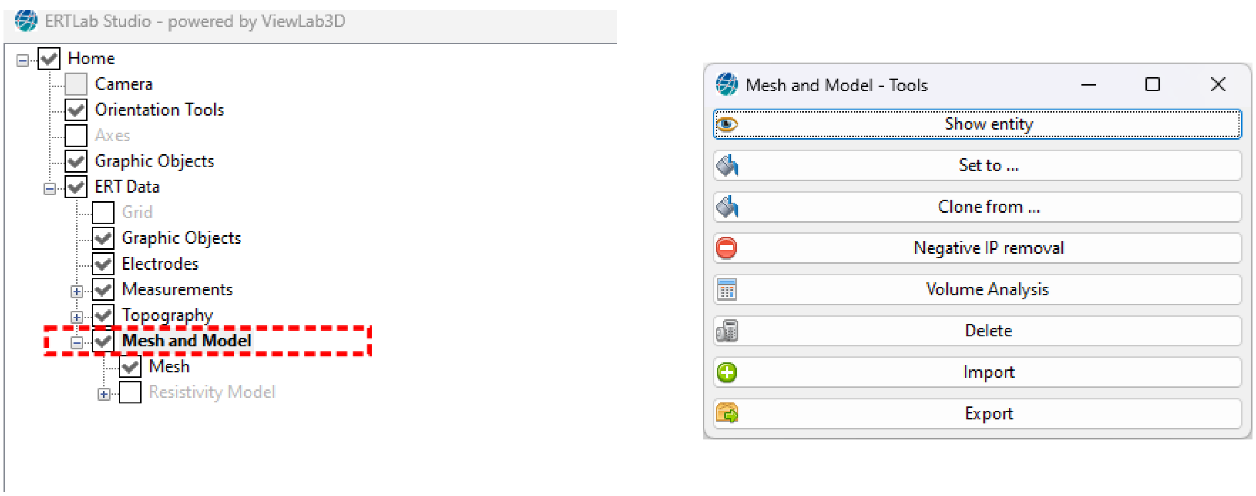


Figure 21.1: Mesh and Model tools panel

All the tools are described in detail in the following sections.

21.1 Show entity

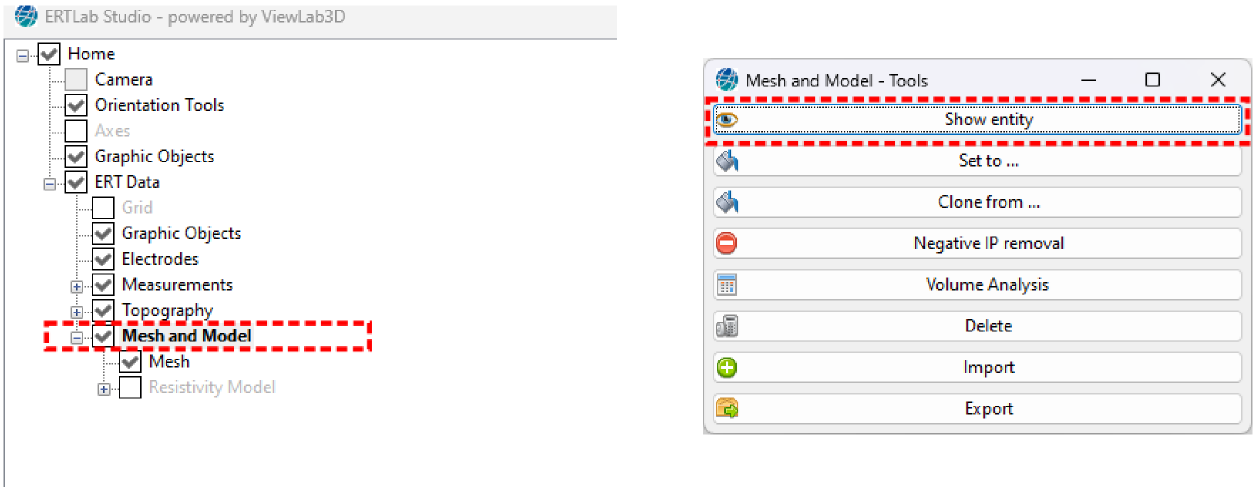


Figure 21.2: Show entity panel

With this command is possible to show hidden entities in the tree, like Resistivity/Conductivity/IP model. By default only resistivity is shown, and the others are hidden.

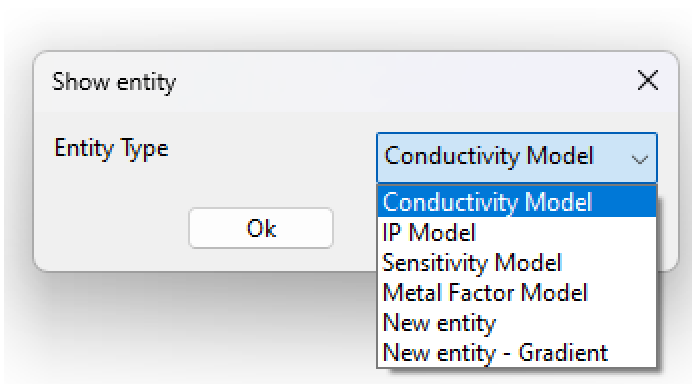


Figure 21.3: Entity selection

Selection a “*New entity*” it will be created a new Model node in the tree, with empty values (that can be filled with the commands *Set to* or *Clone from*).

Selection a “*New entity - Gradient*” it will be created a new Model node in the tree, filled with values computed with “*Gradient*” operator, based on the model that will be selected with the window that will subsequently be shown.

21.2 Set to

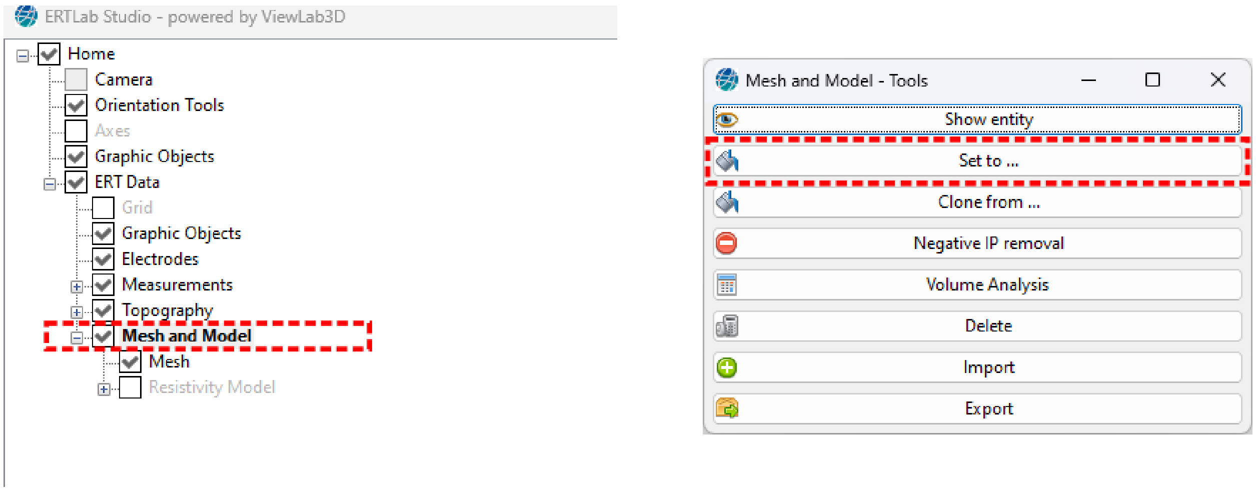


Figure 21.4: Set to panel

This option changes the current value of the Resistivity/Conductivity/IP model. Clicking on *Set to* button opens a new window for selecting the desired *Entity Type* through the pop-up menu (Figure 21.5) and then type the desired value in the *New Value* number box (it is 0 by default). Click on "Ok" to confirm.

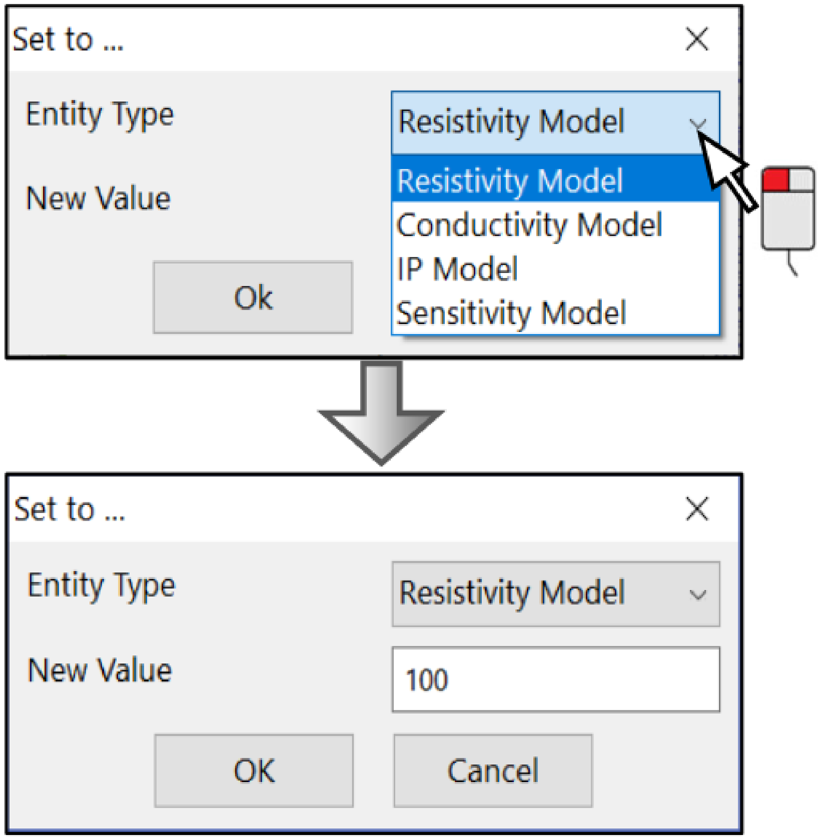


Figure 21.5: Example of Resistivity Model set to 100

21.3 Clone from

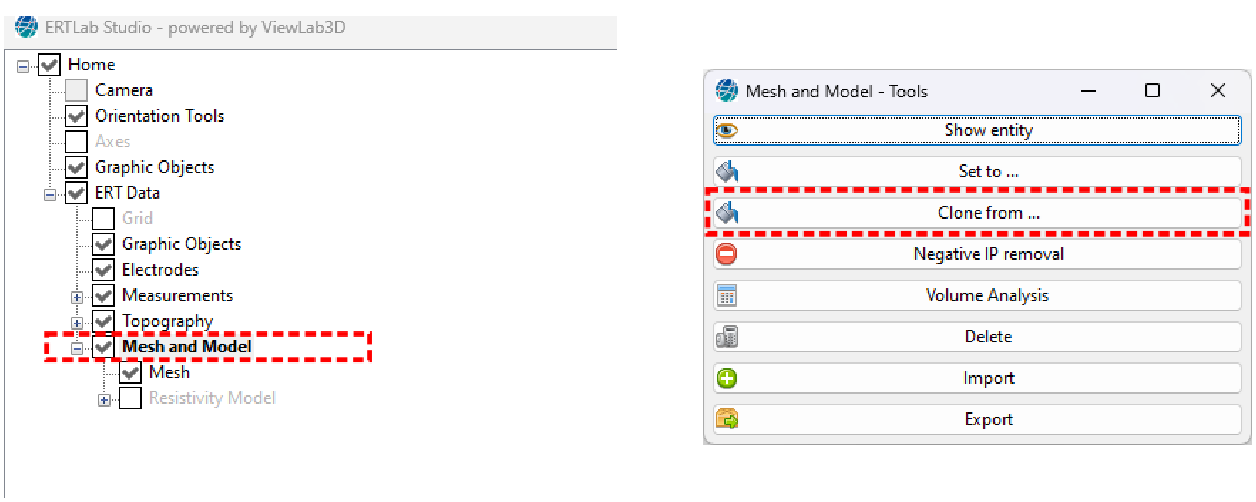


Figure 21.6: Clone from panel

With this command is possible to set in the current model an entity from a different node of the tree. The entities can be like Resistivity/Conductivity/IP model/.... It will be asked to specify the "*Source*" entity (from this or other project opened into the program) and the "*Destination*" entity (in the current project) where to copy the specified model.

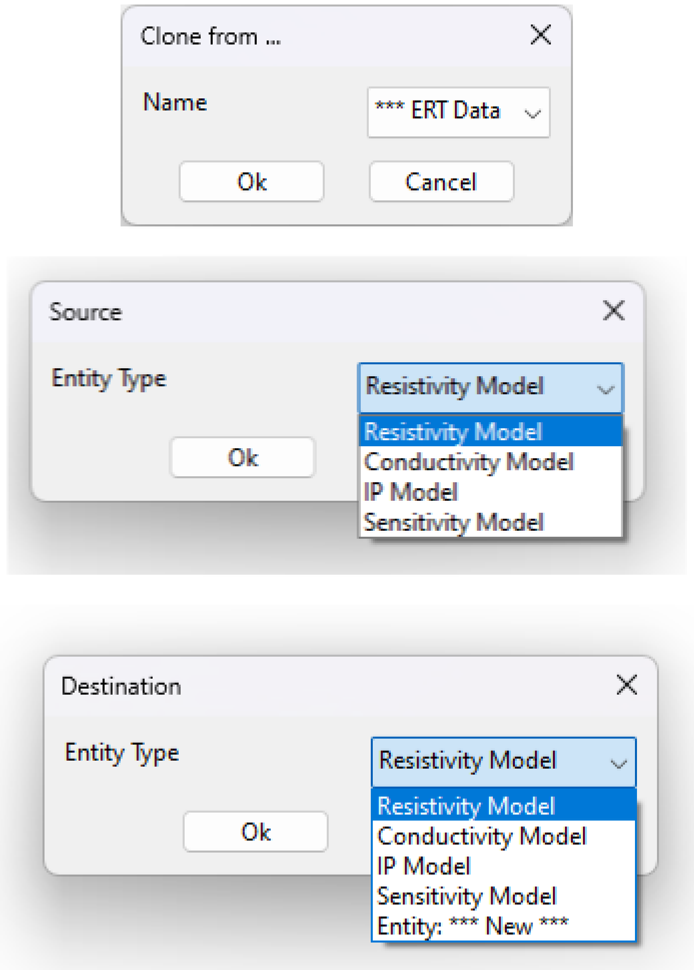


Figure 21.7: Entity selection

If selection as "*Destination*" the "*Entity*" named "*New*" it will then added to the tree a new Model node.

21.4 Negative IP removal

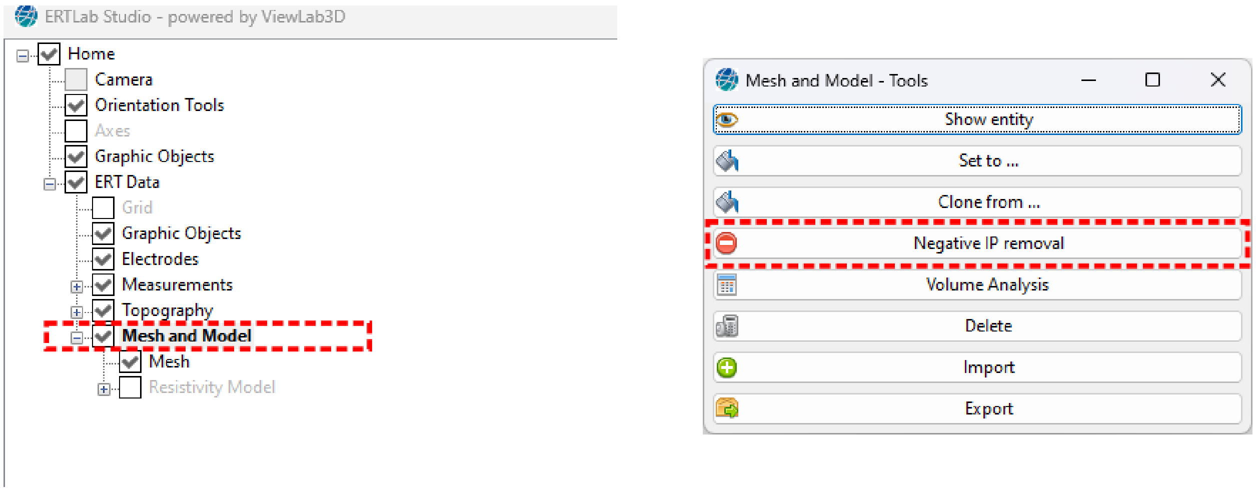


Figure 21.8: Negative IP removal panel

Due to computation reasons it may happen that some negative IP values will occur in the Model after the inversion processes. It is strongly recommended to delete them through the is option. It is important to not confuse these values with negative IP data, which are acceptable and manageable through the proper options (section Histogram)

21.5 Volume Analysis

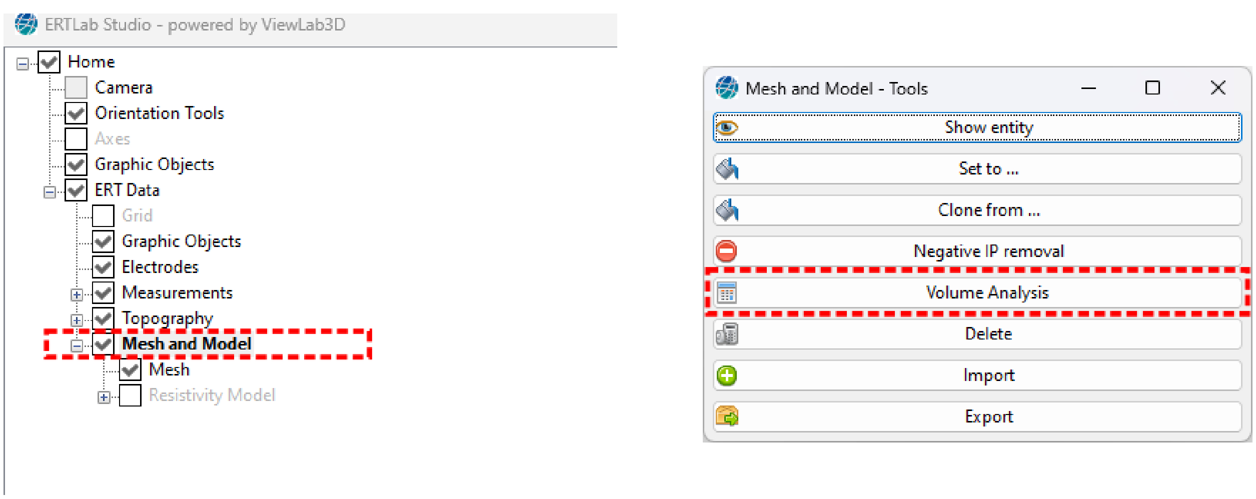


Figure 21.9: Volume Analysis panel

This option allows the calculation of the mesh volume (in m^3 and ‰) and the amount of cells (in numbers and ‰), which have an *Entity Type* within a specified range. In Figure 21.10 the volume is calculated with the resistivity comprised between 50 and 240 Ohm*m, which correspond to 11,113.498 m^3 (28.8‰ of the entire volume) and to 11,308 cells (28.13‰ of the total number of cells).

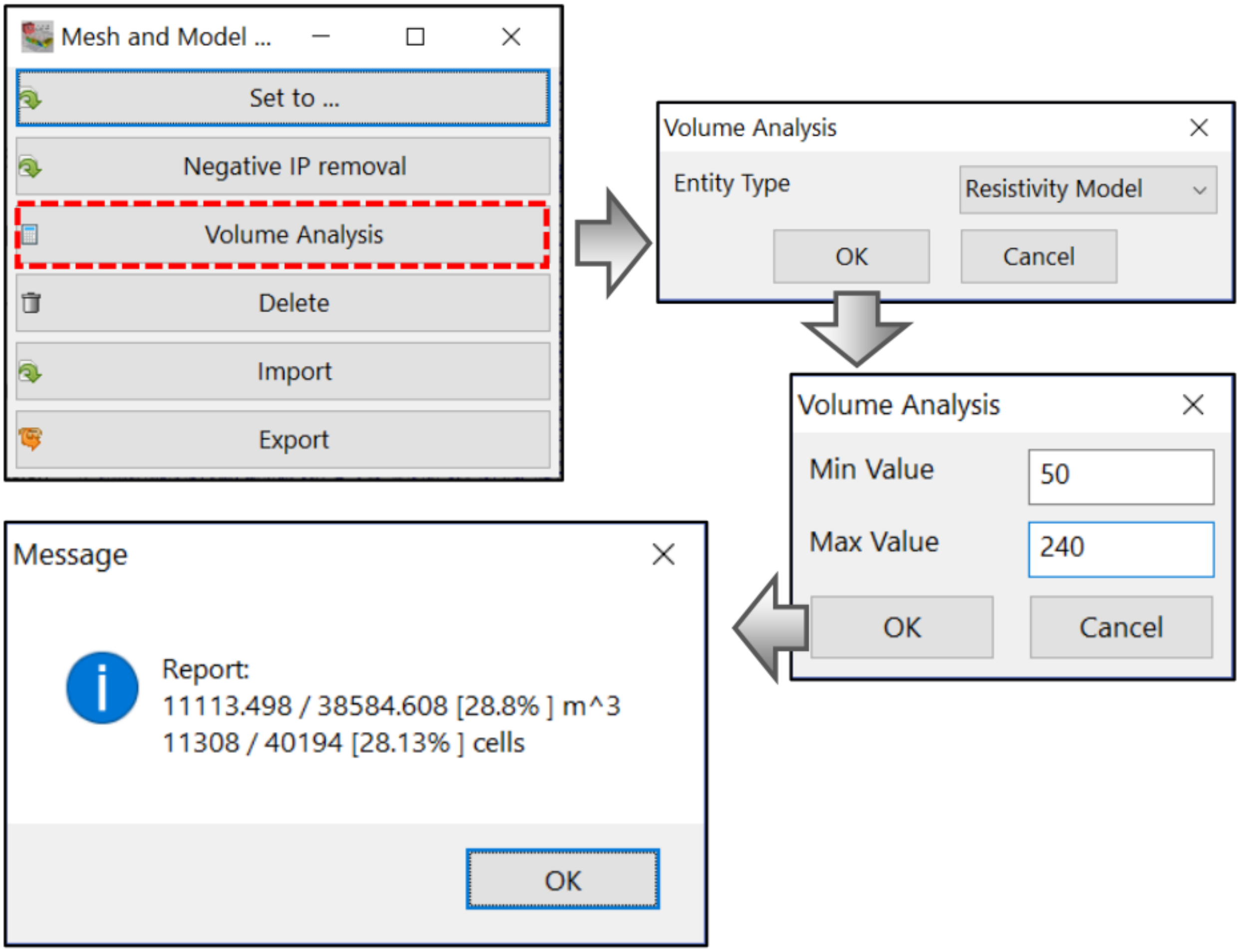


Figure 21.10: Volume Analysis tool

21.6 Delete

Through this option the entire mesh and model project can be deleted (to avoid accidental deletions, a confirmation window will appear).

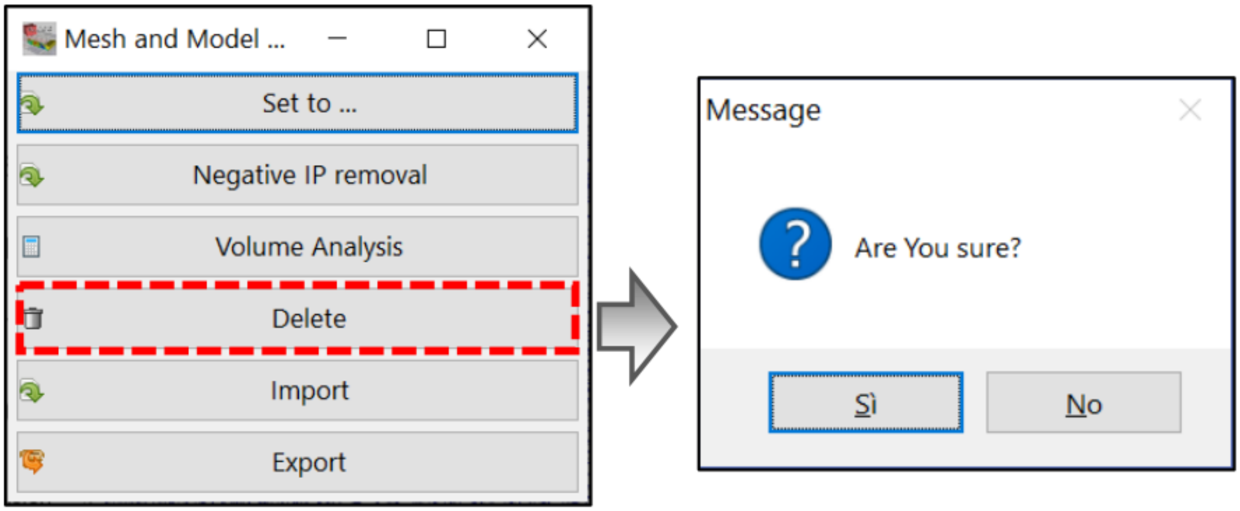


Figure 21.11: Delete tool

21.7 Import

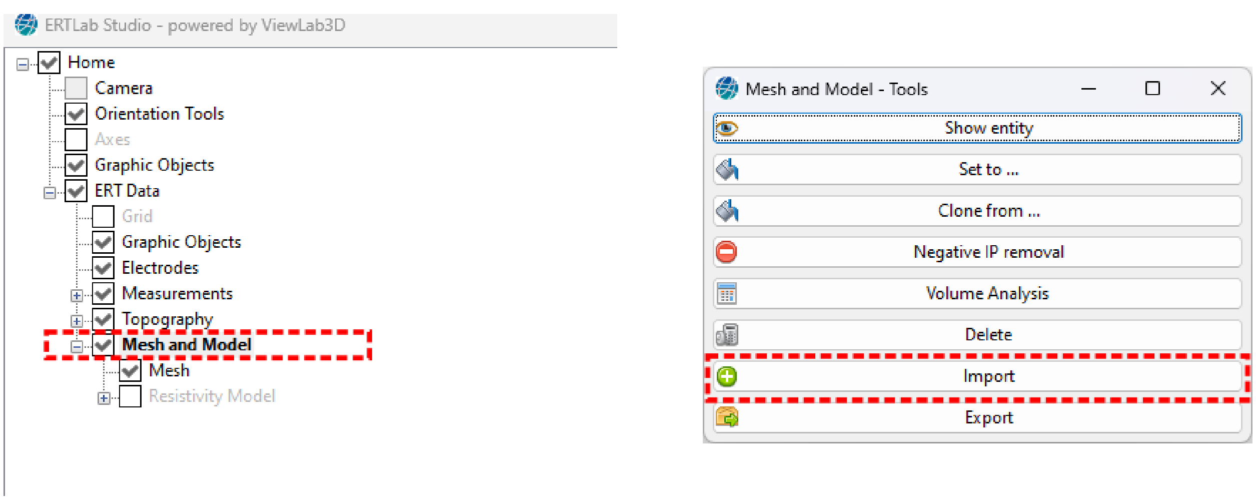


Figure 21.12: Import panel

Use this option to import a Mesh and/or a Model file from a DATA files (*.data or *.wDat or *.txt), a Viewer file (*.vwer), or a XYZ file (*.xyz).

21.8 Export

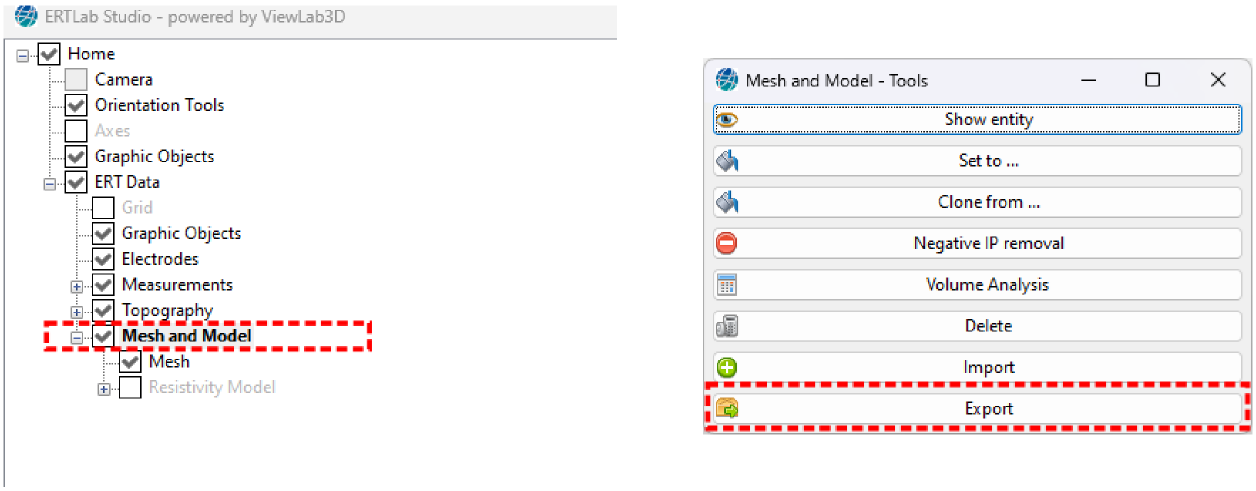


Figure 21.13: Export panel

Use this option to export a Resistivity/Conductivity/IP model, selecting the desired *Entity Type* through the pop-up menu.

If the "*Foreground Only*" is checked, the Background Region is not saved to reduce the necessary space. Note that it is important to export the Background Region also if the output file, after some editing, will be needed to be imported again.

It is possible to export a file as a XYZ file (*.xyz) or as a EVS file (*.xyz or *.fdl).

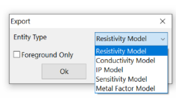


Figure 21.14: Export panel - choice of entity

CHAPTER TWENTYTWO

RUN SEQUENCE GENERATION

By clicking with the right mouse button on the main node of the "*ERT Data*", and choosing "*Run Sequence Generation*", the desired data acquisition sequence can be generated.

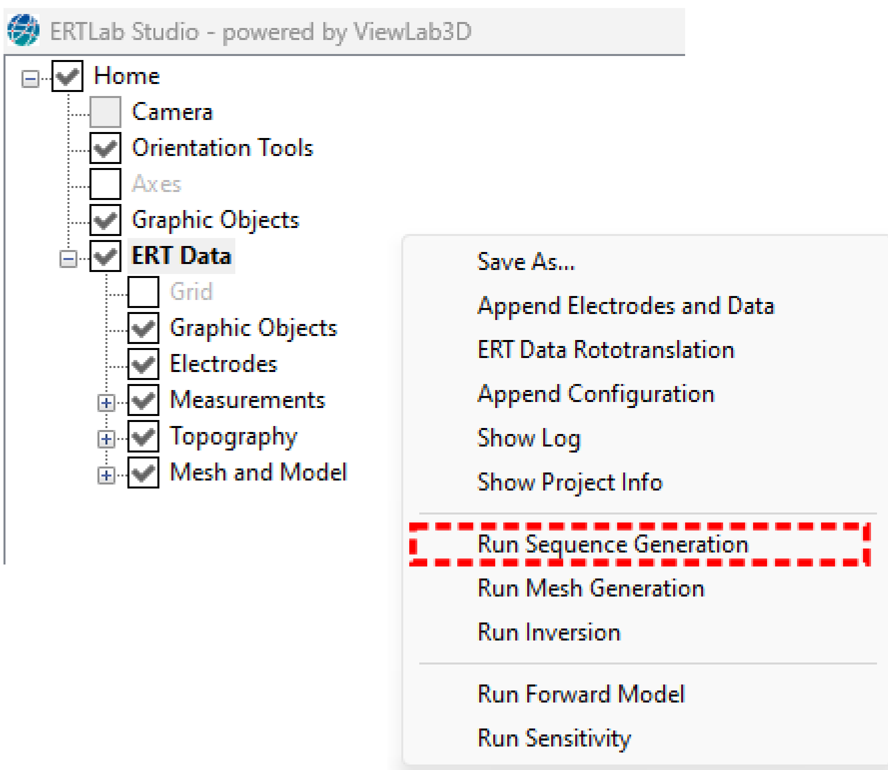


Figure 22.1: Run Sequence Generation

22.1 Select the array type

The first step consists in the choice of the type of array; the main configurations are available (Figure 22.2).

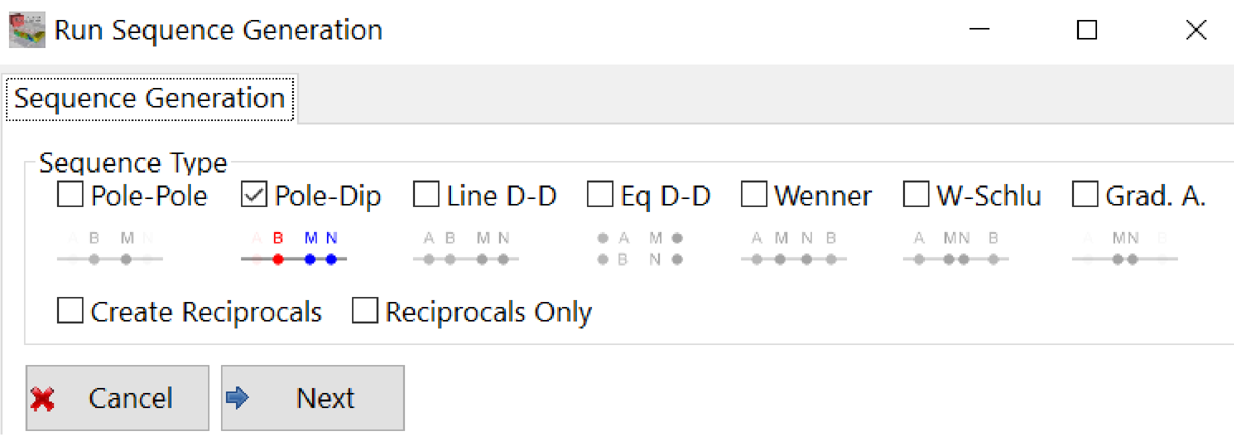


Figure 22.2: Sequence generation, step 1; array definition.

Under each box is a schematic indicating the geometry of the sequences, which are also described below. After having checked the desired sequence type, click a on *Next* button.

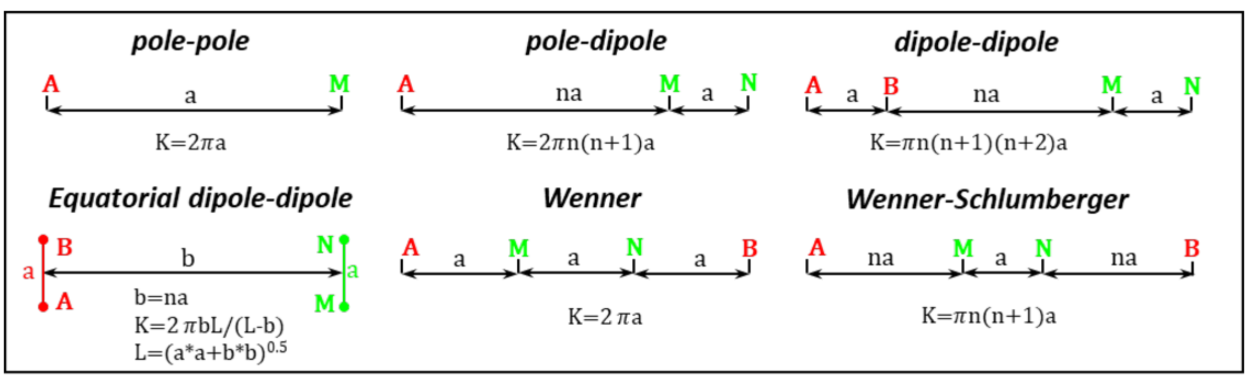


Figure 22.3: Main electrodes array

22.2 Choose array parameters

Once the array type has been selected, the *Next* button will open a window for the choice of the array parameters (Figure 22.4).

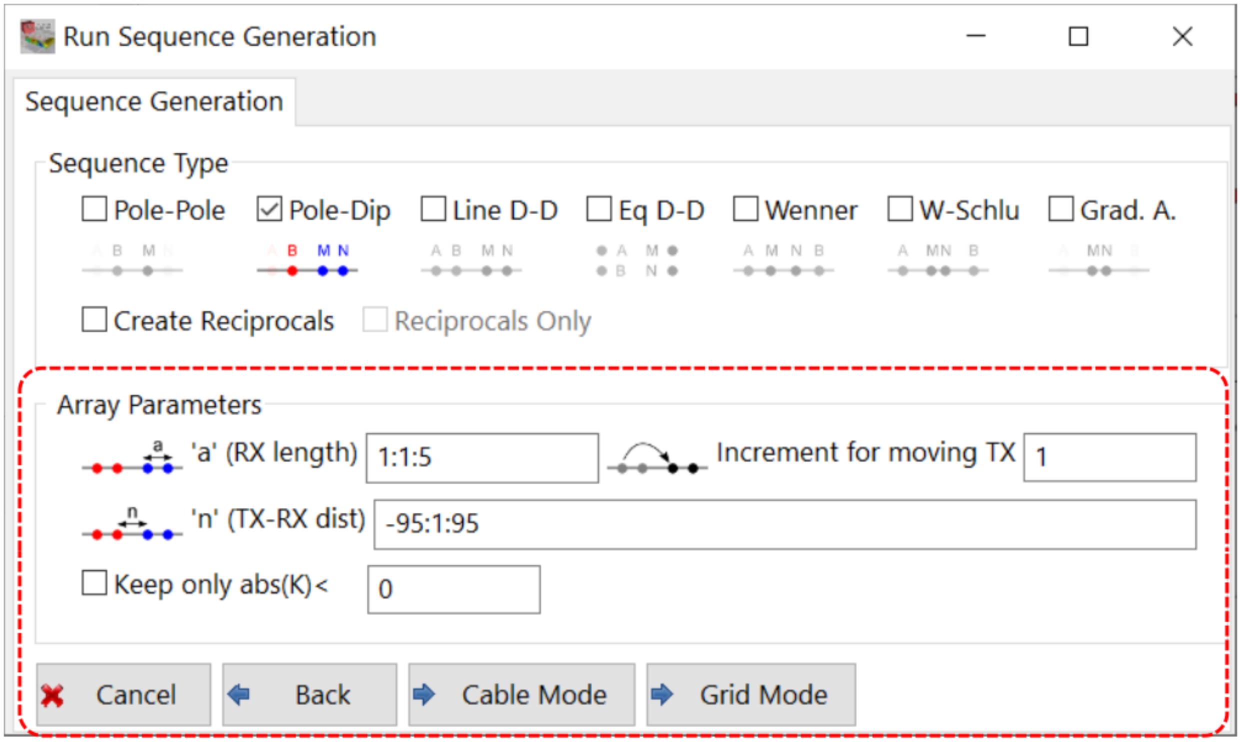
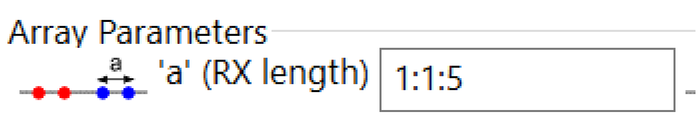
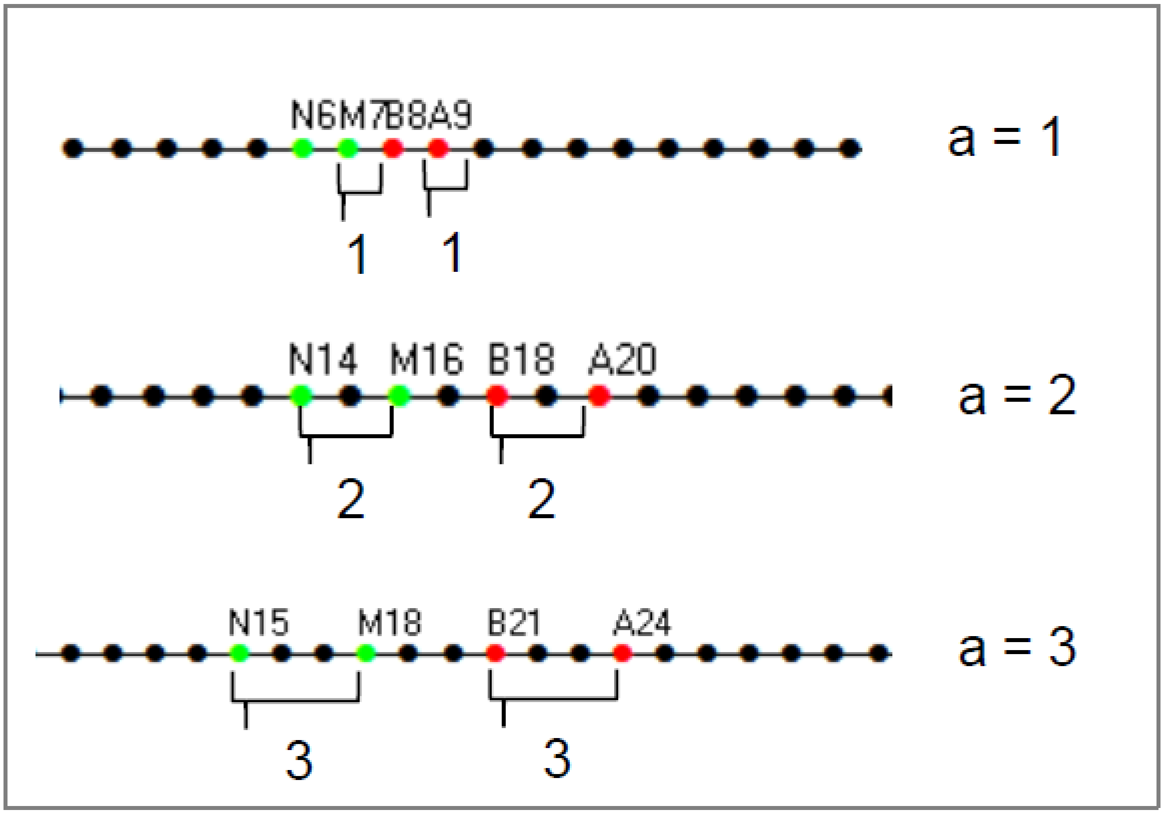


Figure 22.4: Sequence generation, step 2; set array parameters

- Set the *length of the RX-dipole* ("a"), by applying the Mathematical Notation.

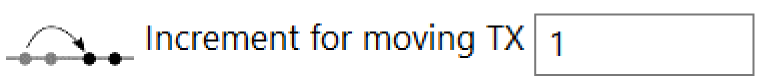


In the example, 1:1:5 the numbers are the *starting value* : *increment value* : *end value*, resulting in the values 1,2,3,4,5.



The values are reported as electrode distances, so 1 means a length equal to the length of the electrodes spacing, 2 a length equal to 2x the length of the electrodes spacing, 3 a length equal to 3x the length of the electrode spacing, etc.

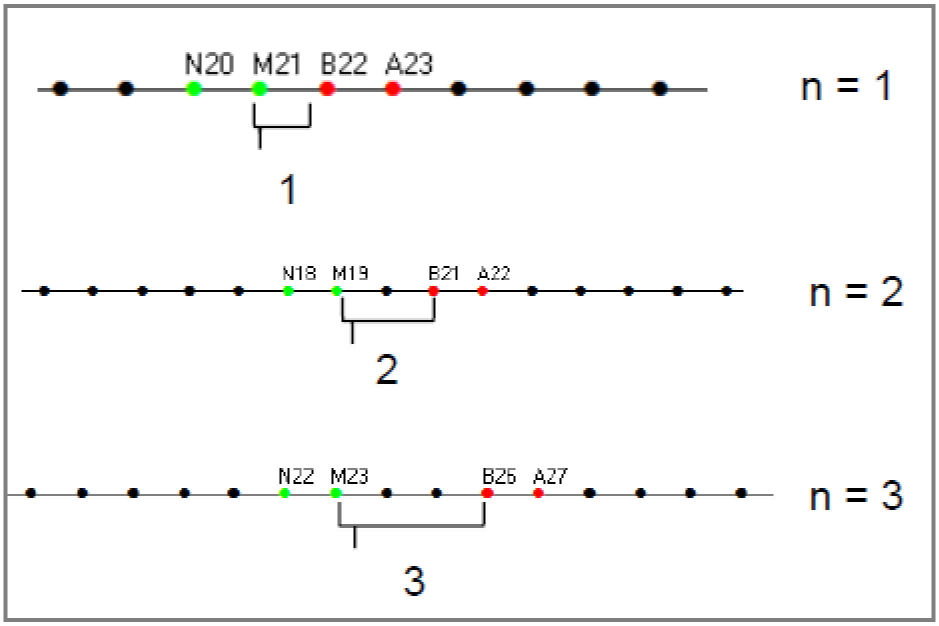
- Set the *increment for moving TX*.



This value is expressed in units of electrodes distance as well and is set to 1 by default.

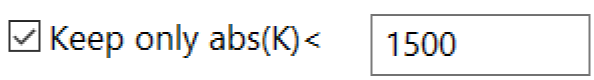
- Set the *distance between RX-dipole and TX-dipole* ("n"), by applying the Mathematical Notation.

- In the example, -48:1:48 are the *starting value* : *increment value* : *end value*, resulting in the values -48,-47,-46,-45, ….,0,1,2,3,……47,48.



The values are also reported in dipole distances (take-out distance), so 1 means a length equal to the length of the TX-RX dipole, 2 a length equal to 2x the length of the TX-RX dipole, etc.

- *Constrain the generation of quadrupole* to geometric factor. If the options is selected, the quadrupoles are generated only if the geometric factor K is less than the entered value. This parameter is used to limit the number of measures and avoid the generation of quadrupole with low signal/noise ratio (that is often related to a quadrupole with high "n" value having a long distance between TX and RX). The geometric factor threshold can be for example set to 1500 times the electrode spacing to remove many poor quadrupoles.



22.3 Select Insertion Mode

There are two methods for the generation of sequences:

- Cable Mode
- Grid Mode

To switch to one insertion mode to the other, click on the "*Back*" button (in yellow in Figure 22.5) and choose the desired tool.

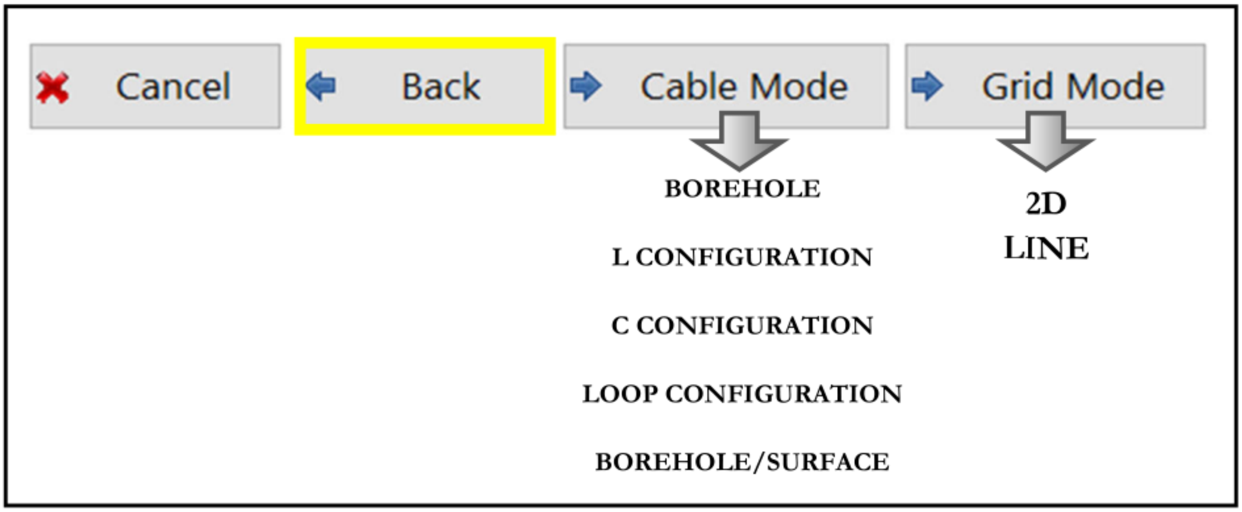


Figure 22.5: Choice of sequence mode generation

22.3.1 Cable Mode

It generates the sequences for cross-cable mode, having transmitters and receivers on different cables.

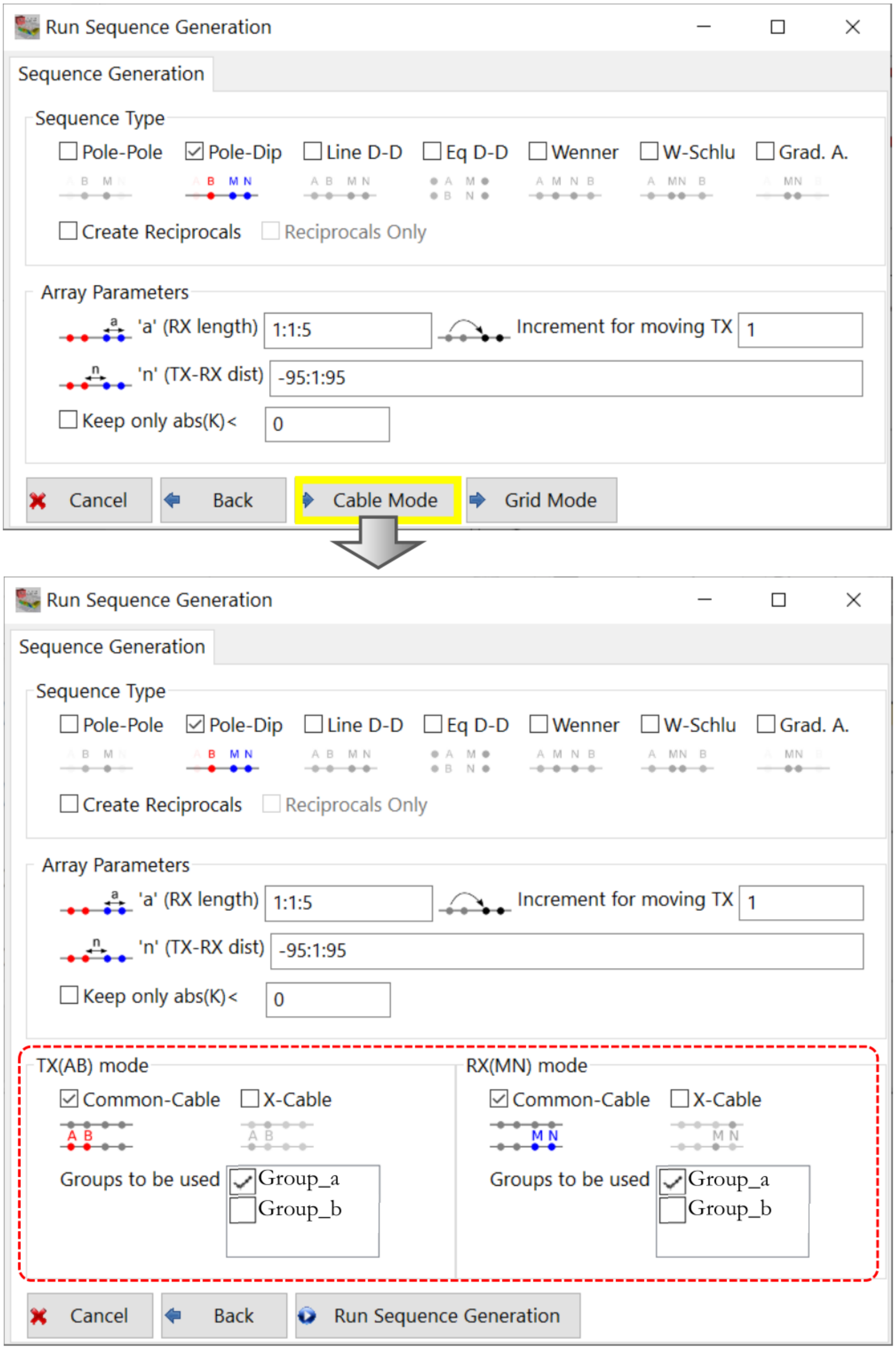


Figure 22.6: Cable-mode insertion panel

It lets the user manage borehole acquisition, surface acquisition with L or C configuration and borehole-surface acquisitions. The panel is separated in TX cable tools on the left and RX cable tools on the right. In both cases it is possible to choose to put the TX or RX in the same cable (*Common-Cable* selection box) or in separated cable (*X-Cable* selection box). Figure 22.7 shows an example for a TX dipole. The RX dipole can be configured like this as well.

Figure 22.7: Example of Common Cable/Cross Cable TX

The Group on which to put the RX and the TX is selectable at the bottom of the window checking the appropriate box (in Figure 22.8 two cable are load in the project, Group_a cable and Group_b.)

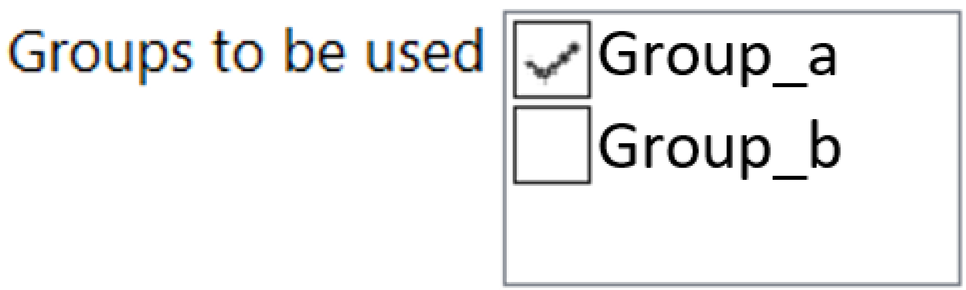


Figure 22.8: Groups where to put TR/TX

Clicking on *Run Sequence Generation* button generates the electrode sequence specified and a confirmation message will appear when the generation is completed (Figure 22.9).

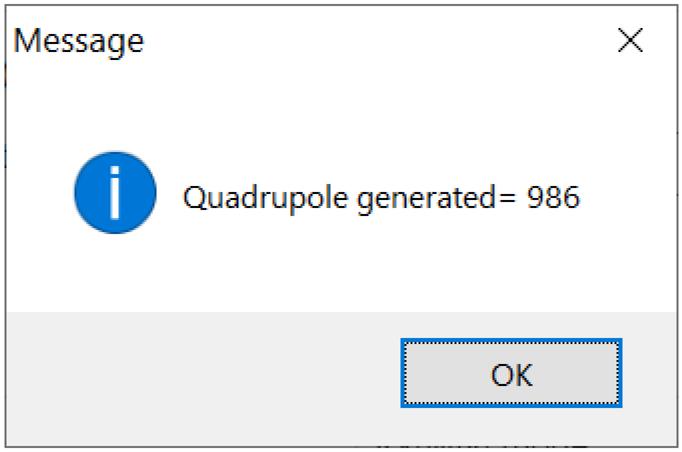


Figure 22.9: Quadrupole generated information message

Starting again from the beginning of the sequence generation procedure and creating a new sequence. A new message will appear, to choose adding the new sequence to the previous, or deleting the older and keep only the last sequence generated. In either case a new message will indicate the resulting numbers of new quadrupole generated.

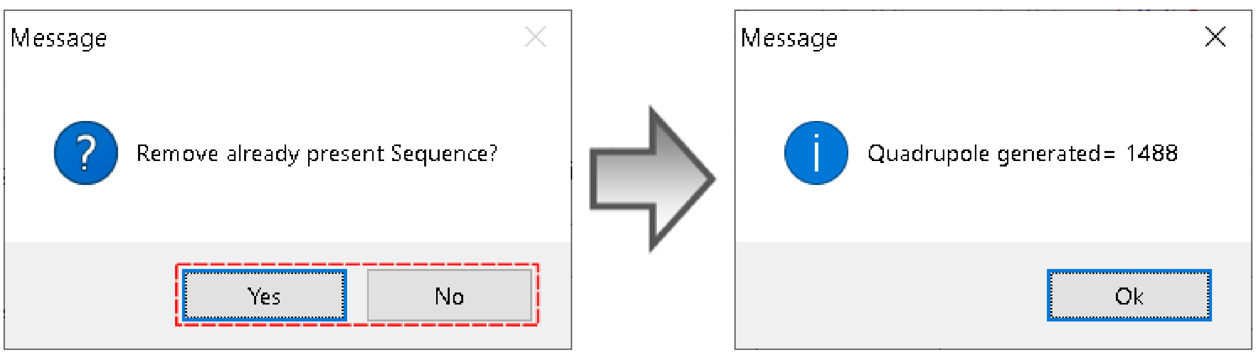


Figure 22.10: Panel to choose if remove or not the previous sequence

22.3.2 Grid Mode

It generates the sequences for 2D surface profiles or 3D grids.

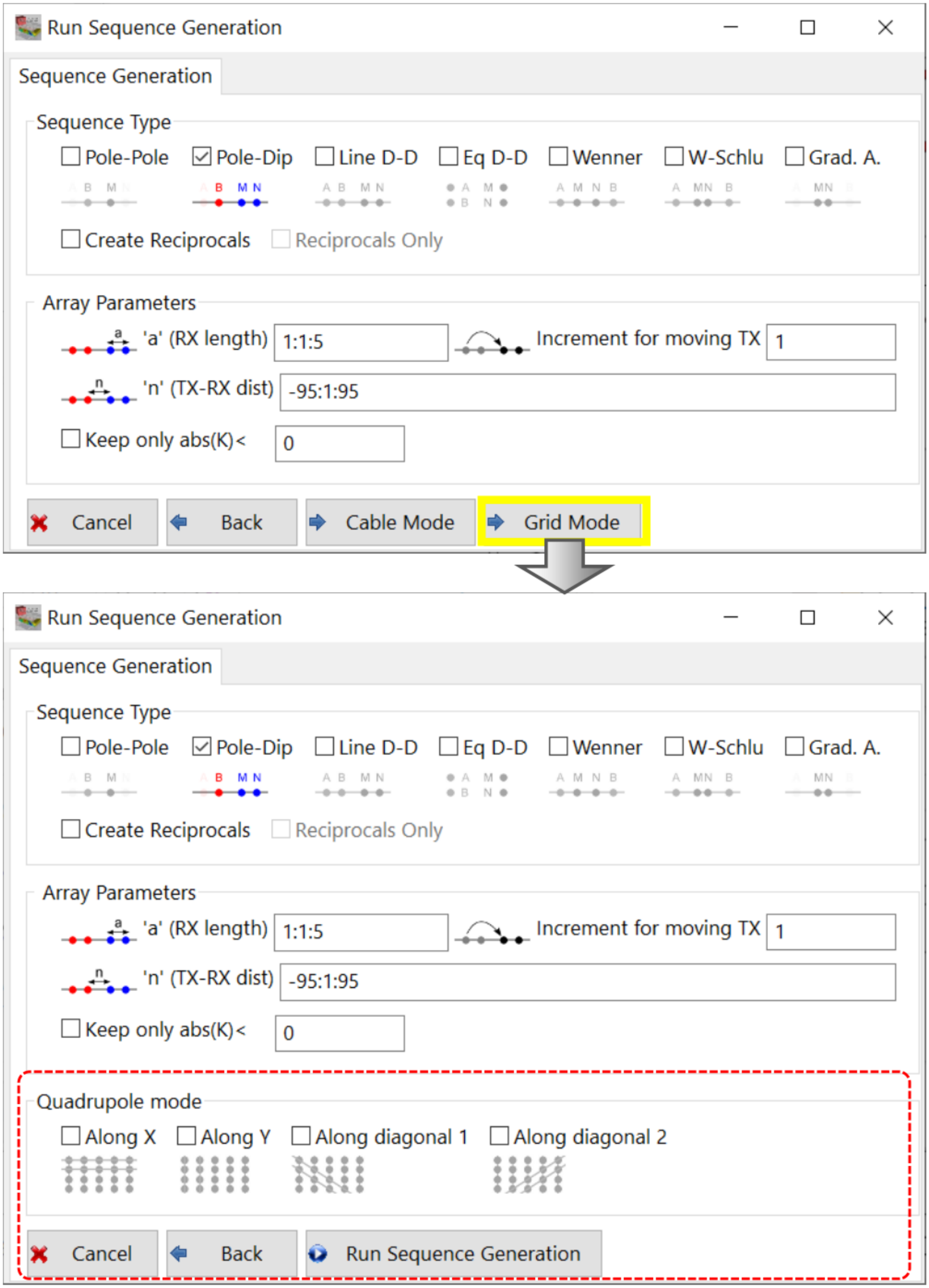


Figure 22.11: Gird-mode insert panel

Through this panel it is possible to choose the quadrupole direction, which can be along the main X and Y Axes and in diagonal. As seen in Figure 22.12, each button helps to make the wanted choice.

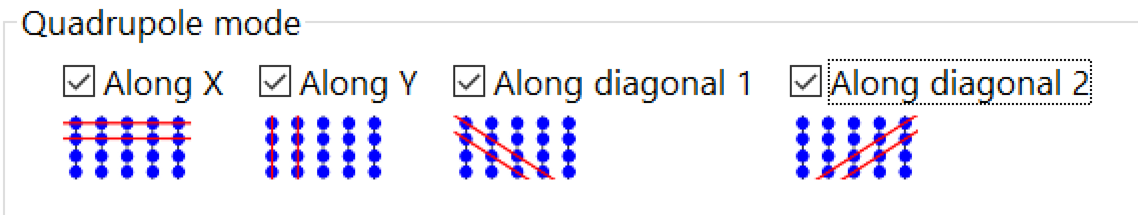


Figure 22.12: Choice of quadrupole acquisition direction

CHAPTER

TWENTYTHREE

RUN MESH GENERATION

By clicking with the right mouse button on the main node of the "*ERT Data*", and choosing "*Run Mesh Generation*" opens a new window that lets the user configure the mesh.

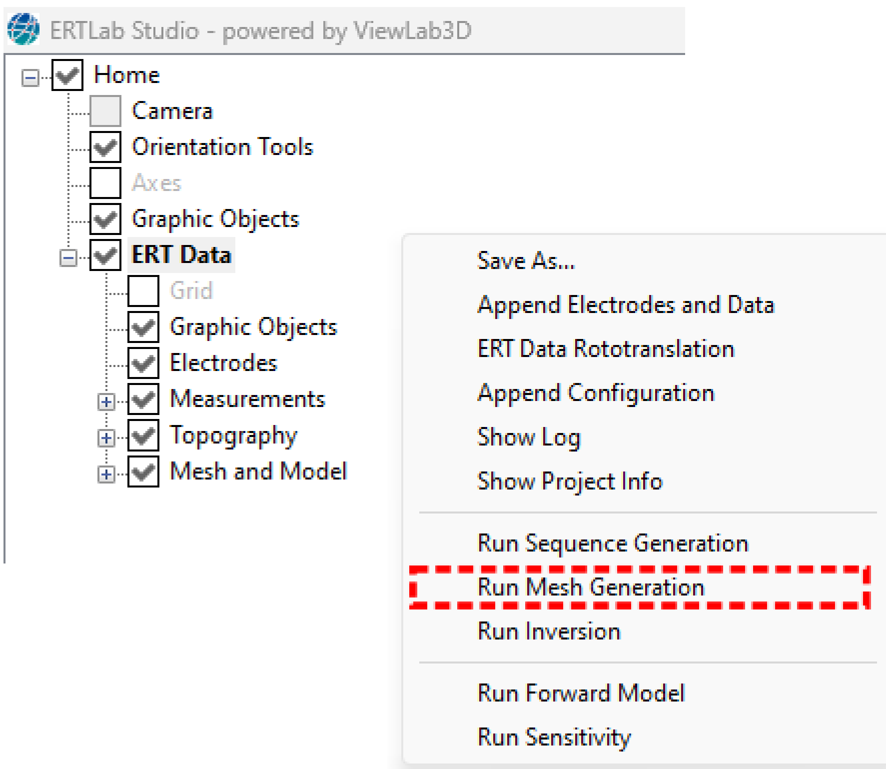


Figure 23.1: Run Mesh Generation

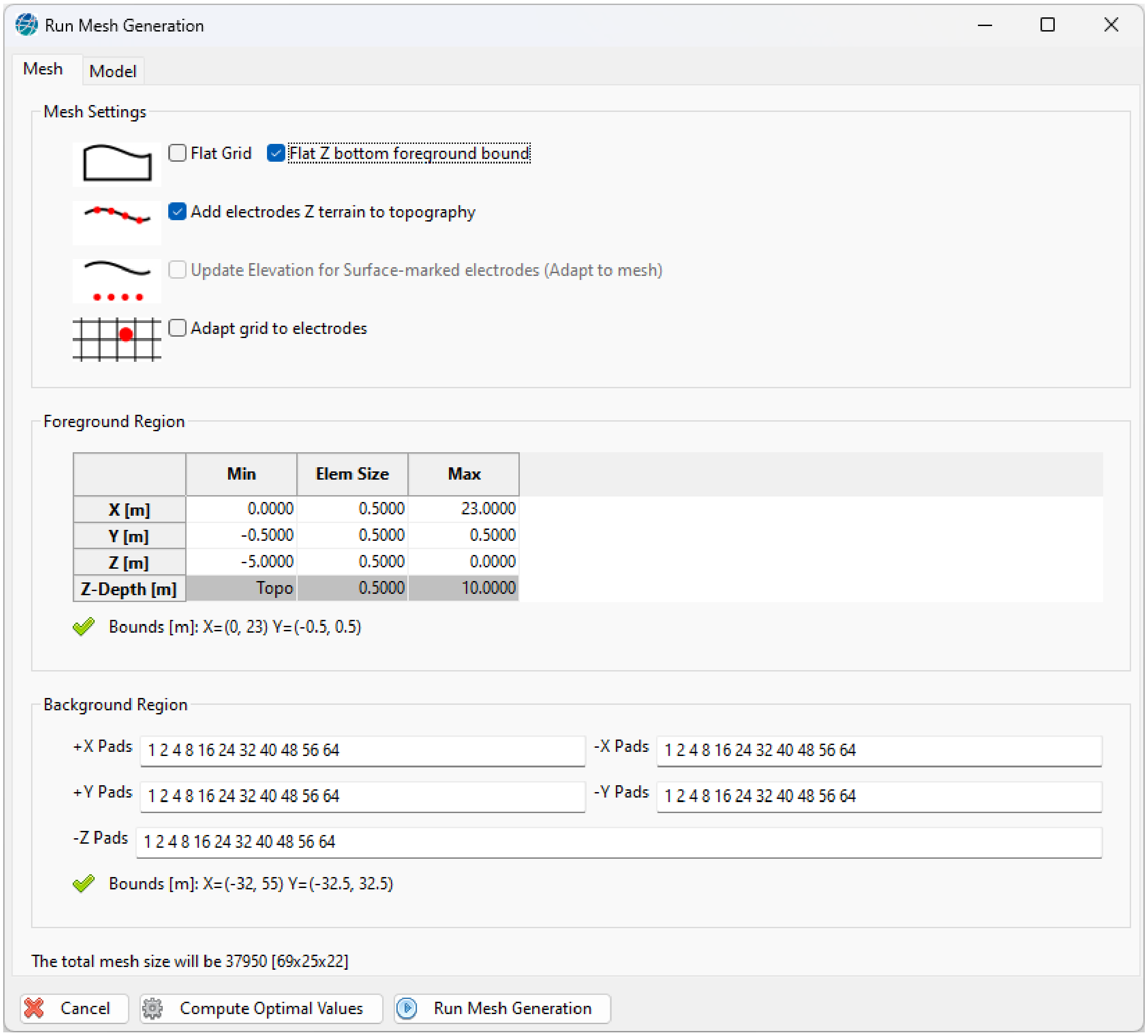


Figure 23.2: Mesh generation panel

The mesh generation window has two tabs: **Mesh** and **Mode**.

The **Mesh tab** is in turn divided into three main areas: *Mesh Setting*, *Foreground Region* and *Background Region*.

23.1 Mesh

23.1.1 Mesh Setting

In this area there are options for setting the mesh in relation to the topography and the positioning of the electrodes in the 3D space.

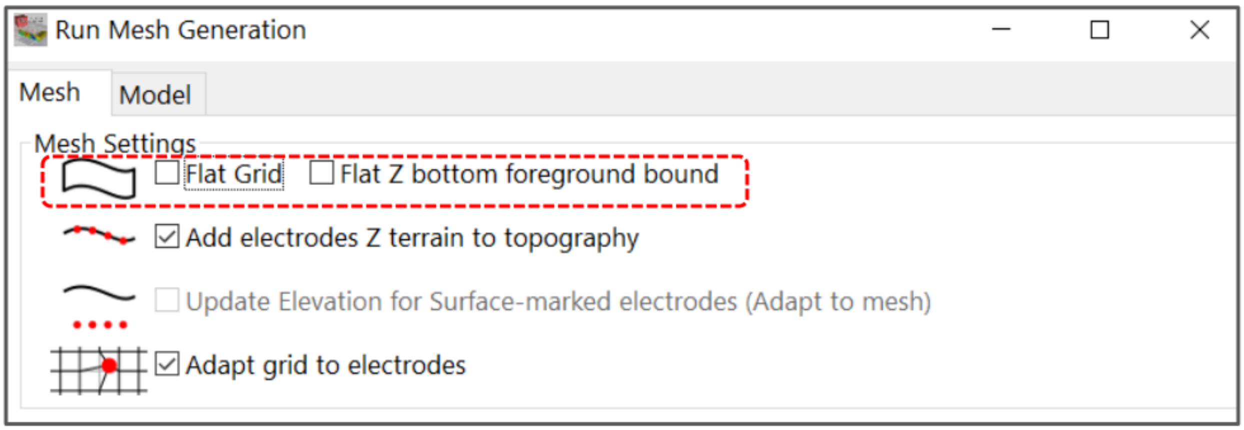


Figure 23.3: Shape of the mesh

If **Flat Grid** is active, the top and the bottom of the mesh surface are flat, so there is no topographic information. Otherwise, if the Flat Grid is inactive, the top of the mesh follows the topography. The geometry of the bottom of mesh is set by "**Flat Z bottom foreground bound**" option. If selected, the bottom of the mesh is flat, otherwise it also follows the topography trend. The icons of the tool changes in function depending on the users selection helping to make the right choice. In summary, there are three possible cases:

- **1° case: Flat Grid selected**:

In this case, the mesh is flat so there is no topographic information.

- **2° case: Flat Grid not selected, Flat Z bottom selected**:

In this case the top of the mesh follows the topography. The volume of the individual cells of the mesh do not change, so there will be parts of the mesh with cells more "stretched" and areas with cells more "squeezed".

- **3° case: Flat Grid not selected, Flat not Z bottom selected**:

In this case both the top and the bottom of the mesh follow the topography, so the thickness of the mesh is constant, and the cells have all the same size. These three cases are illustrated in Figure 23.4, where there is a simple synthetic case on the left and a more complex real case on the right.

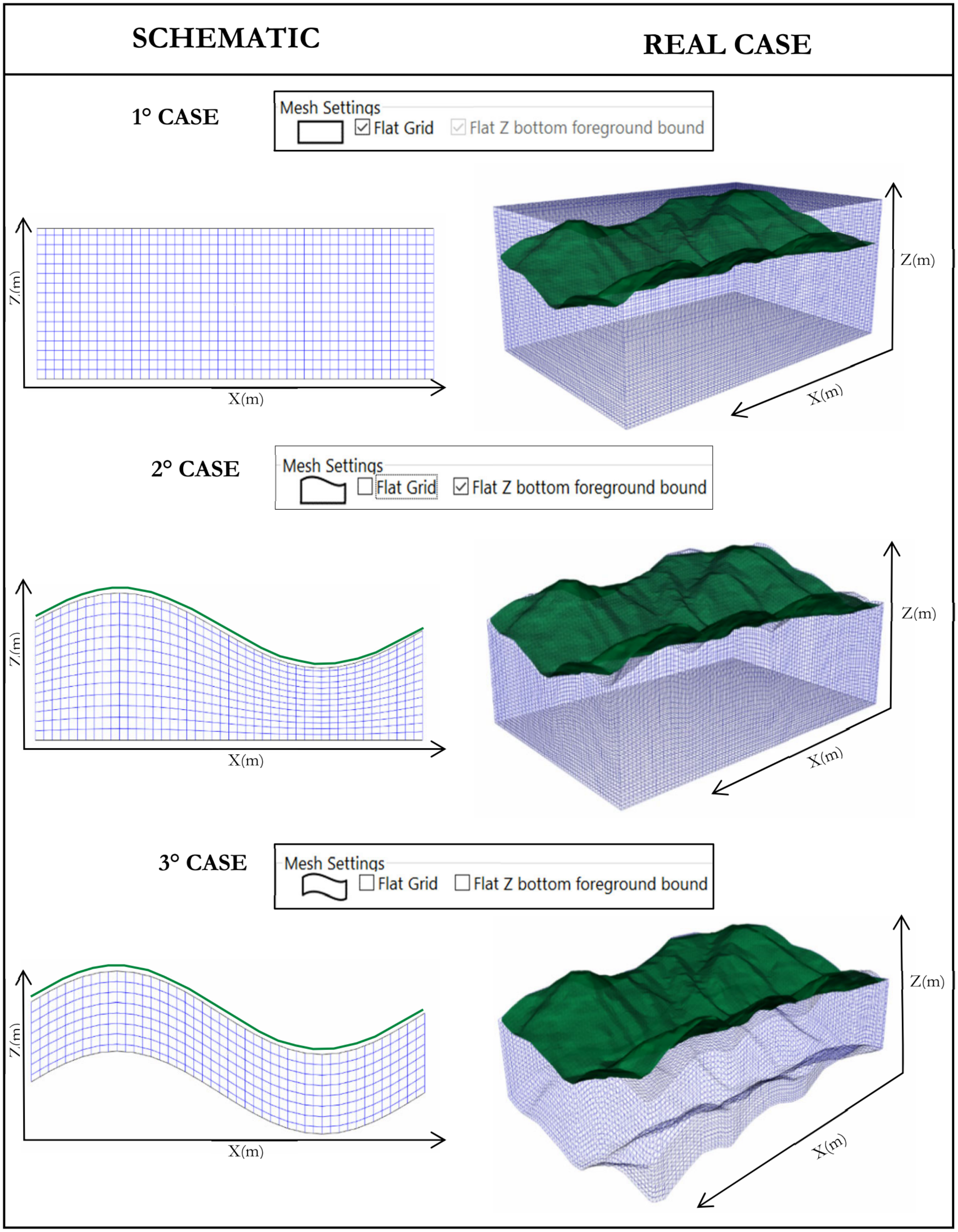


Figure 23.4: Examples of different kinds of mesh

If the mesh has the topographic information (previous case 2 and 3), it may happen that the z coordinates of one or more electrodes are not consistent with the topography file. In most cases, the Z information of the topography is more accurate than the Z value of the electrodes. To adapt the electrodes to the mesh you can select the appropriate check box:

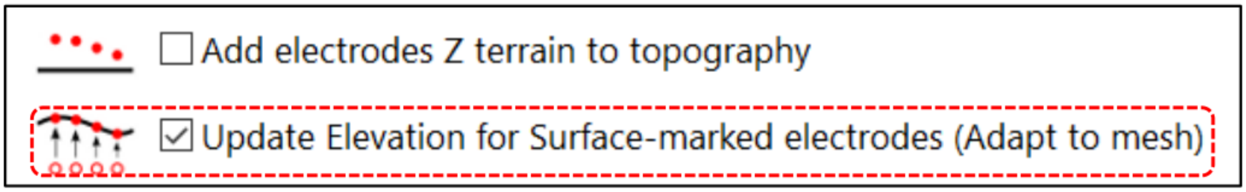


Figure 23.5: Adapt electrodes elevation to mesh

If for some reason the Z of the electrodes must be taken into consideration, e.g. if the topography file is not a DEM (Digital Elevation Model) and just composed of some scattered points, it is possible to add the electrode Z terrain to the topography file, through the dedicated check box:

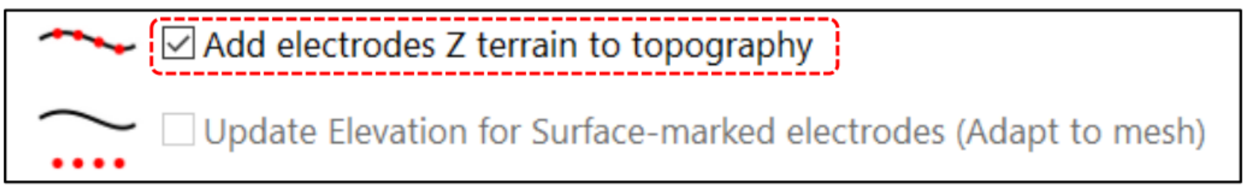


Figure 23.6: Add electrodes elevation to topography

Figure 23.7 shows a schematic example, with an unrealistic gap between the information of the Z value of the electrodes and the Z of the topography, The electrodes follow a sinusoidal shaped curve and the topography is instead flat. In the left case the "*Add electrodes Z terrain to topography*" is selected, so the electrodes remain fixed and the mesh changes shape to adapt to the Z electrodes trend. If "*Update Elevation for Surface marked electrodes (Adapt to Mesh)*" is selected, the mesh remains flat and the electrodes move to its surface (on the right in Figure 23.7).

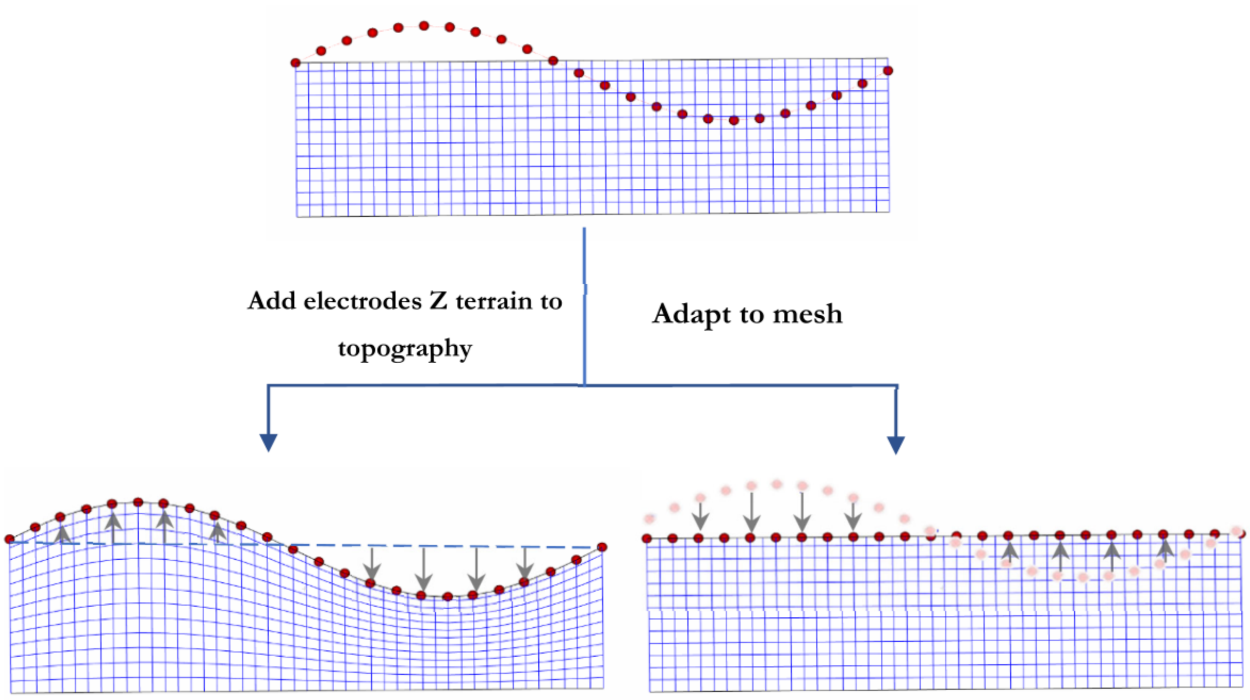


Figure 23.7: Examples of different kind of mesh in relation to the rule of the Z electrodes information

If there is a topography file loaded, selecting "*Add electrodes Z terrain to topography*" is deduced from the Z electrodes coordinates information. The last tool of the *Mesh Setting* tab manages the position of an electrode in relation to the node mesh position, in the XY plane.

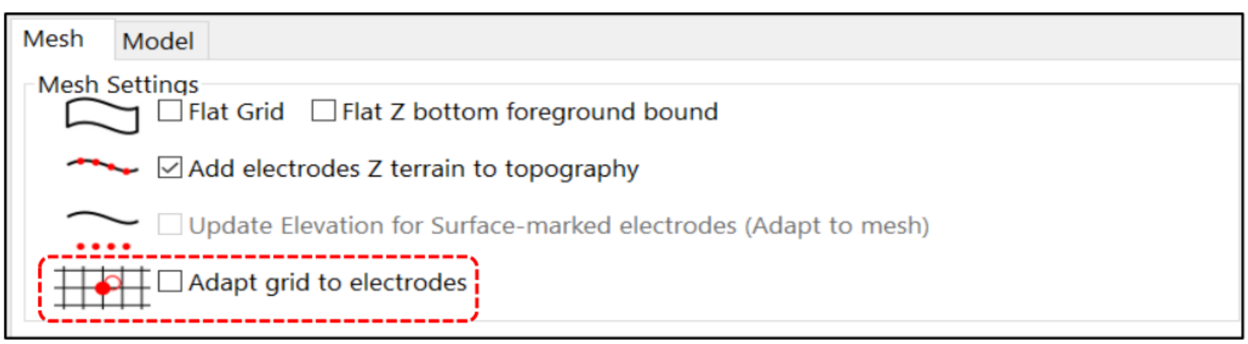


Figure 23.8: Adapt grid to electrodes

In fact, it is possible that the position of one or more electrodes does not coincide with any node of the mesh, as in the following example (Figure 23.9). In this case it is possible to distort the mesh so that the electrodes position coincide with its nodes.

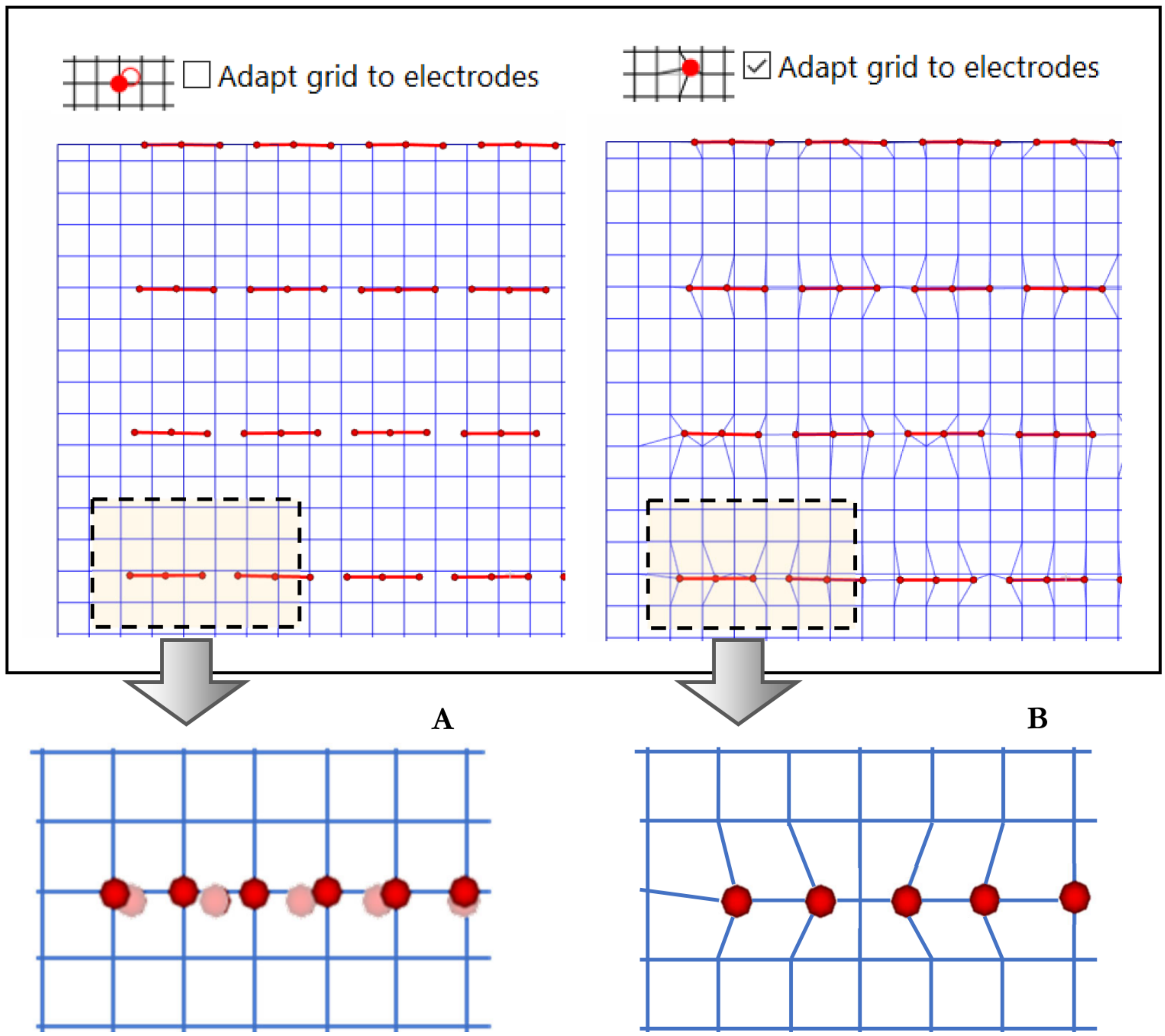


Figure 23.9: In A, “Adapt grid to electrode” is not checked so the electrodes not coincident with the node moves on the nearest node; in B “Adapt grid to electrode” is flagged so the mesh is deformed to adapt the node to the position of the electrodes, which does not change position

23.1.2 Foreground Region

The mesh is divided into two main areas, the background and the foreground region. The **Foreground Region** (red area in Figure 23.10) is the portion of the mesh which includes the investigated area and is defined by the geometry of the electrodes on the ground; outside is the **Background Region** which defines an area necessary for the mitigation of the boundaries effects, theoretically infinite.

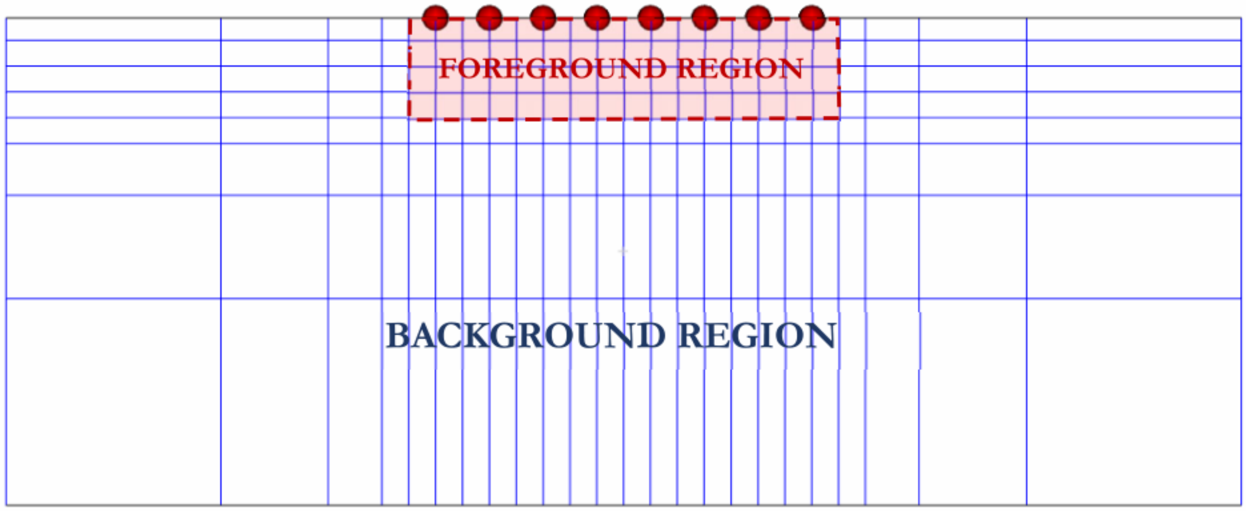
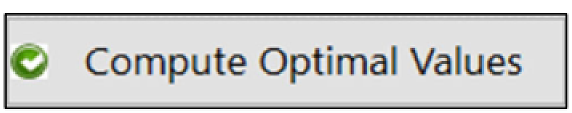


Figure 23.10: Background and Foreground regions

It is possible to set the values of the *Foreground Region* through the dedicated table show. By default, the limits values are the same as the maximum and minimum electrodes coordinates and the size of the cell is half the minimum electrodes distance of the data set. To obtain these default values click on "*Compute Optimum Value*" button. Doing so, the software will compute the more appropriate values for the table, basing on the electrodes spacing.

Foreground Region

	Min	Elem Size	Max
X	0.0000	0.5000	7.0000
Y	0.0000	0.5000	5.0000
Z	-1.4337	0.5000	0.0000
Z-Depth	Topo	0.5000	1.4337



However, it is possible to change these values typing the desired numbers in the table. It is advisable to check the *Element Size* dimension because it may be excessively small. This can happen if for logistical reasons two electrodes were placed in the ground at a distance significantly lower than all the other electrodes, the software will consider half of this minimum distance as the size of the cells of the entire mesh. For example, if all the electrodes are at 2 m distance of each other except

2 electrodes, which are at 0.4m of distance, the correct cell size value is 1m and not 0.2 m as the software suggests (Figure 23.11).

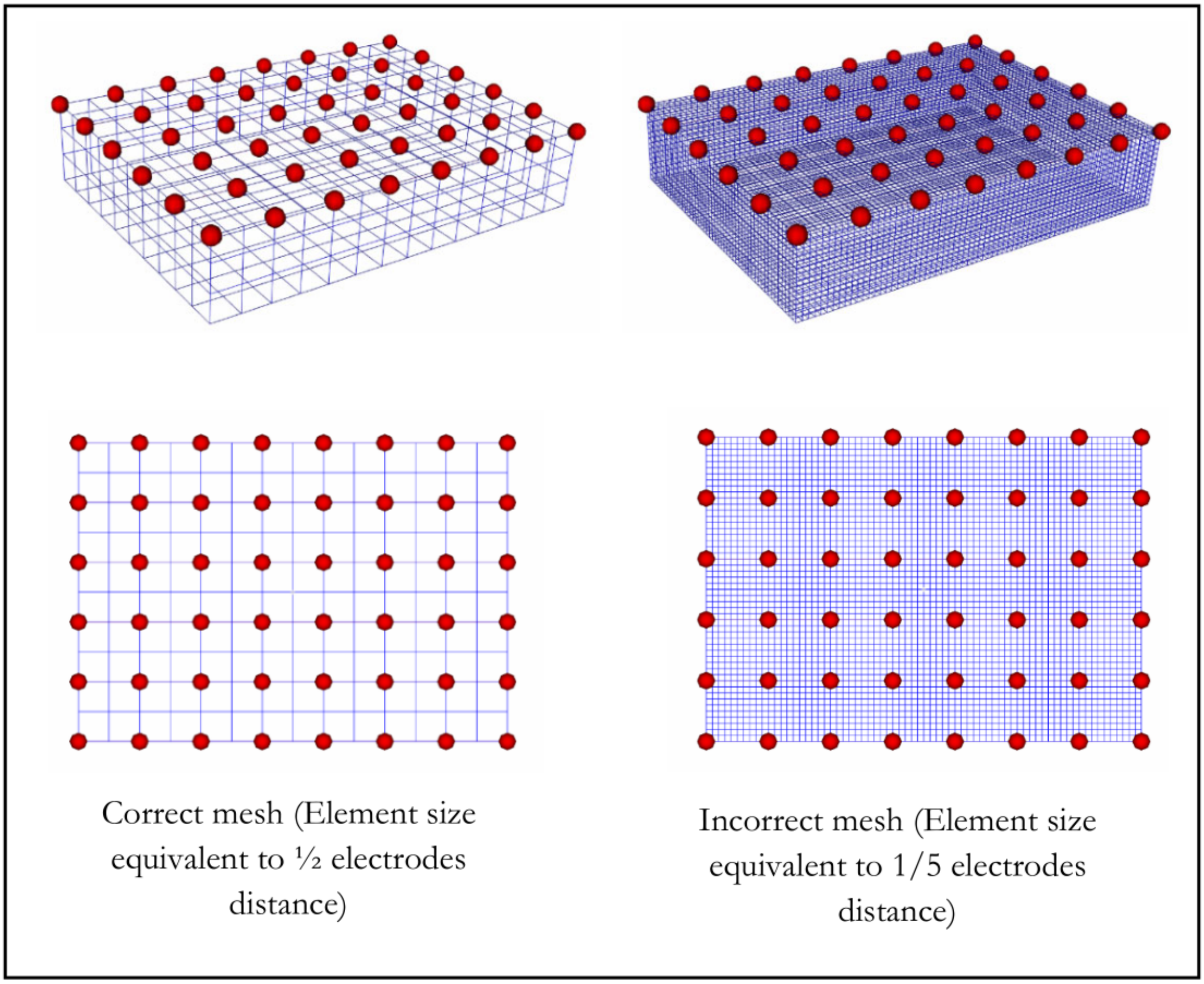


Figure 23.11: Example of two mesh with different cell sizes

If the bottom of the mesh is flat, the *Z-Depth* is not editable and the corresponding row is greyed out, because there is not a constant thickness only a minimum and maximum value of the Z (the positive values are above the surface and the negative underground). Alternatively, if the bottom of the mesh follows the topography the thickness is constant, so the *Z-Depth* is editable (Figure 23.12).

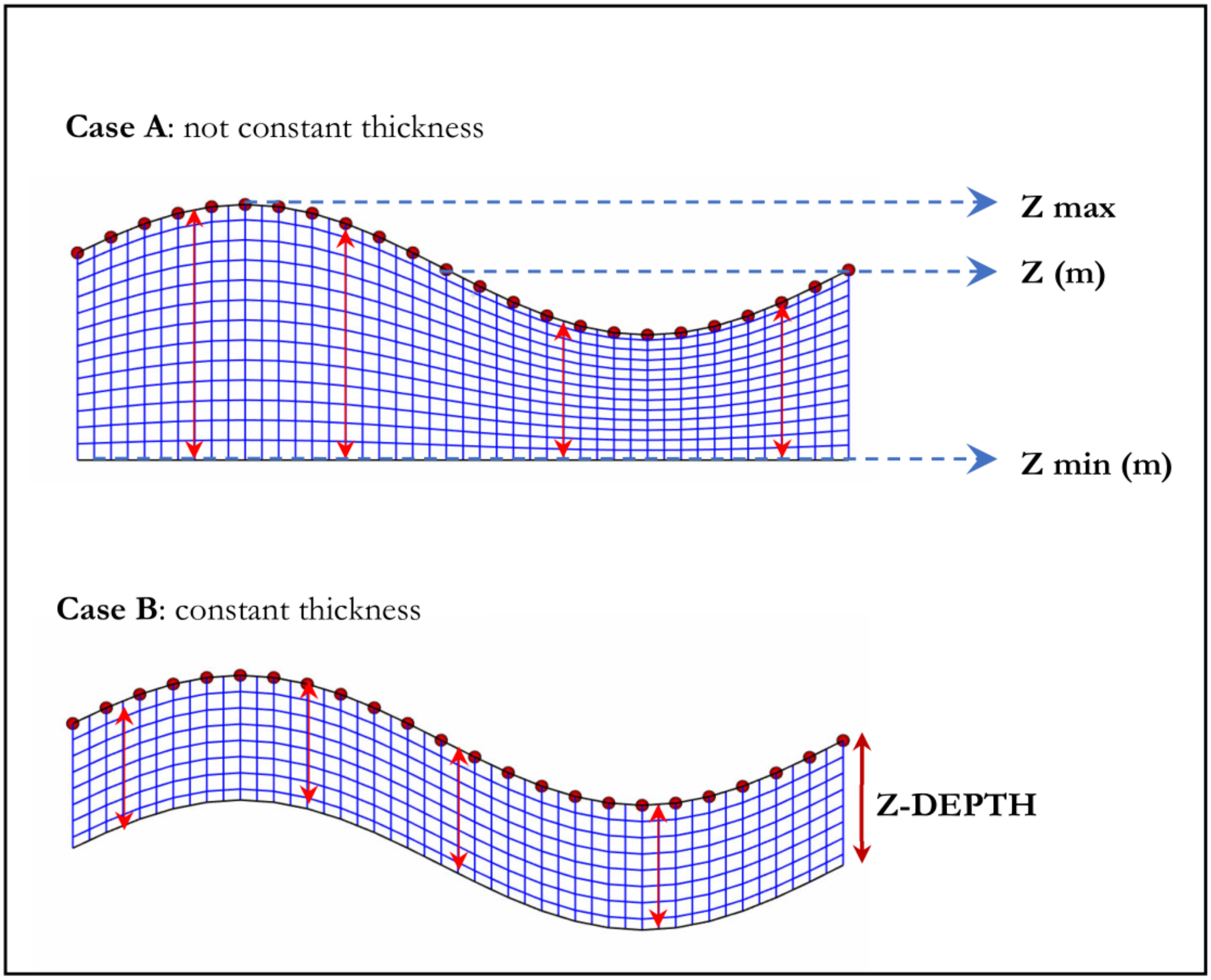
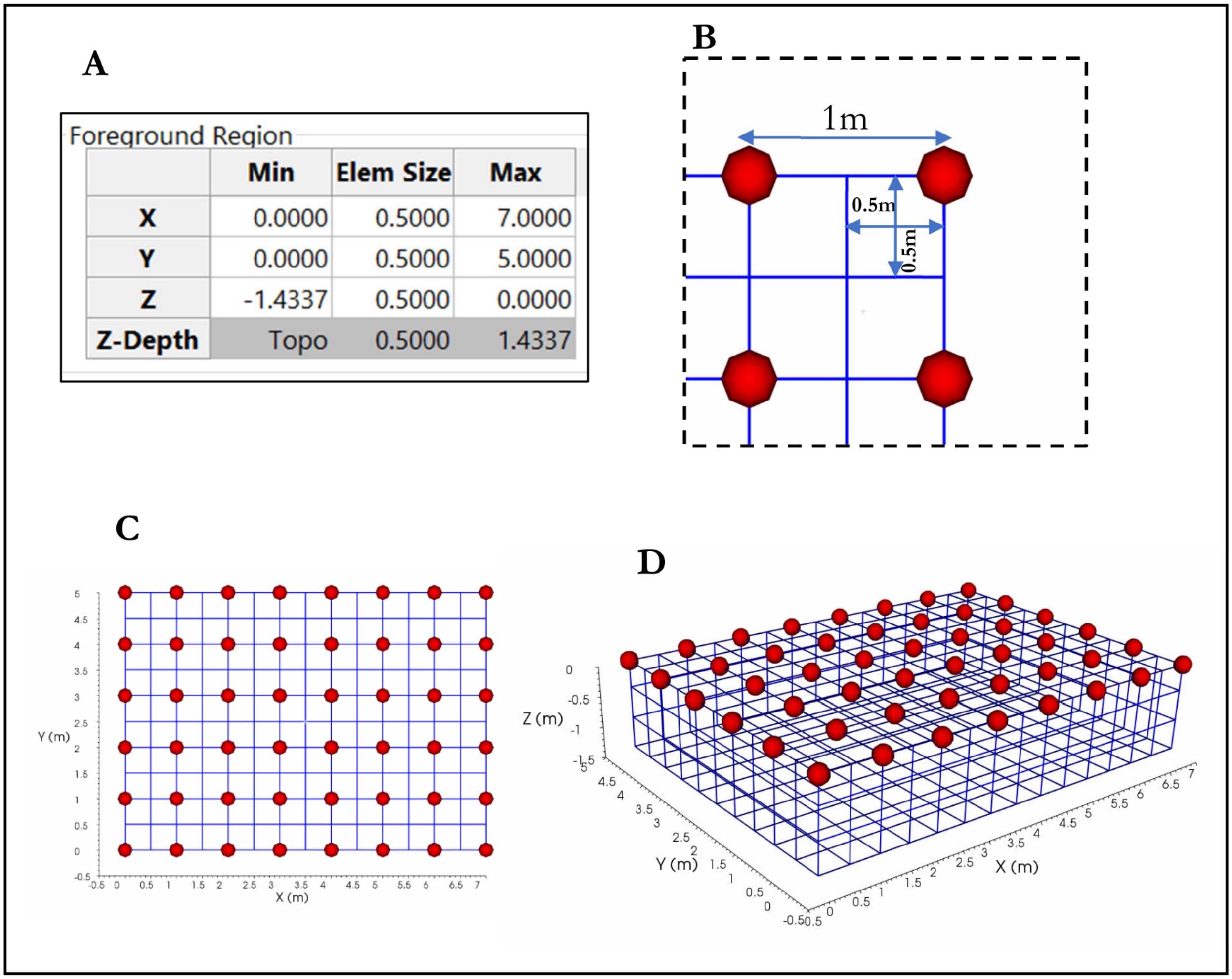


Figure 23.12: Mesh_Z parameter setting

In Figure 23.13 an example is shown of a *Foreground Region* generated by the values reported in the table; the distance between electrodes is 1m, so the size of the cells is 0.5m.

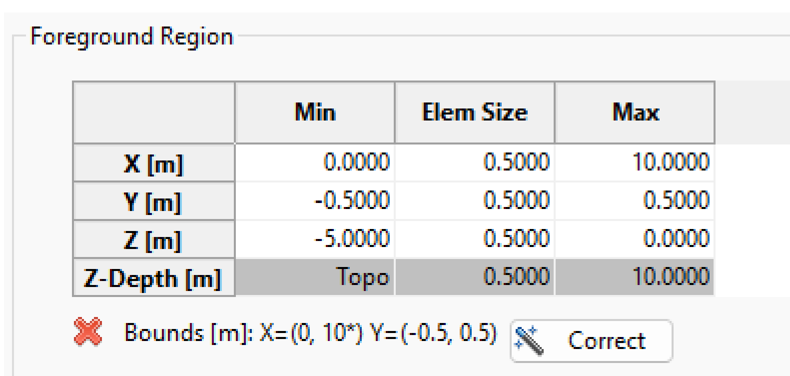


Foreground Region

	Min	Elem Size	Max
X	0.0000	0.5000	7.0000
Y	0.0000	0.5000	5.0000
Z	-1.4337	0.5000	0.0000
Z-Depth	Topo	0.5000	1.4337

Figure 23.13: In A Foreground Region table. In B detail of the mesh with the size of the cell. In C e D the Foreground Region of the mesh in two different points of view

If the current settings of the Foreground Bounds does not include all the electrodes a warning is shown, and pressing the button "*Correct*" it is possible to be helped with an automatic adjustment.



Foreground Region

	Min	Elem Size	Max
X [m]	0.0000	0.5000	10.0000
Y [m]	-0.5000	0.5000	0.5000
Z [m]	-5.0000	0.5000	0.0000
Z-Depth [m]	Topo	0.5000	10.0000

Bounds [m]: X=(0, 10*) Y=(-0.5, 0.5) Correct

Figure 23.14: Foreground Bounds Warning

If the mesh can be rotated to optimize the total number of cells a message will be shown, and pressing the button “*Correct*” it is possible to accept the suggestion. Then, because after the Mesh Generation the rotation performed does not need anymore, it is suggested to restore the previous coordinates rotating back the dataset.

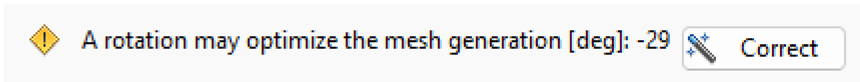


Figure 23.15: Rotation suggestion

In the upper part of Figure 23.16 an example of a mesh generated without any optimization on the rotation. It can be seen in the lower part of the image that after an appropriate rotation the electrodes will lay on the X axis so the mesh will better fit them, it will follow as final result that the Mesh size will be smaller than before.

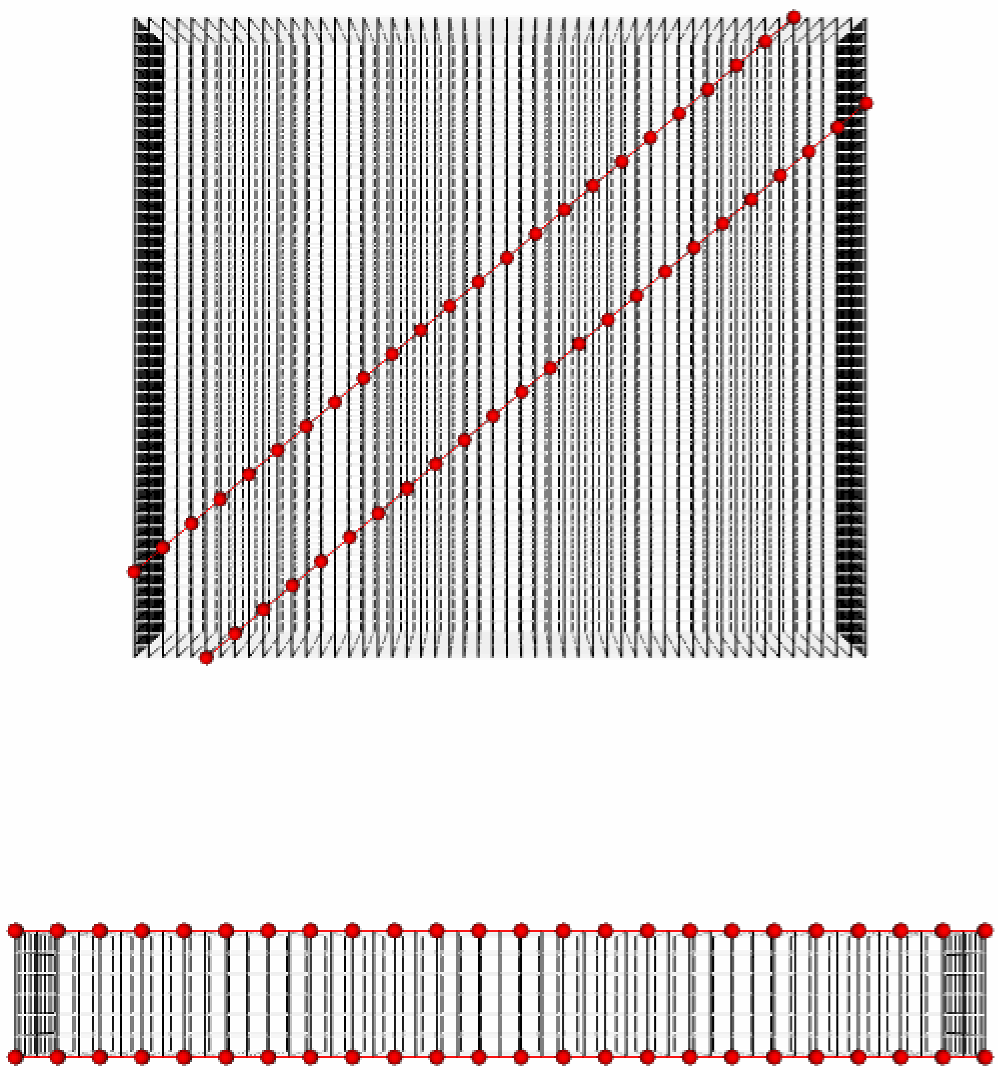


Figure 23.16: Example of rotation optimization

23.1.3 Background Region

The **Background Region** is the portion of the mesh which is outside the *Foreground Region* and defines an area necessary for the mitigation of the boundary's effects, theoretically infinite.

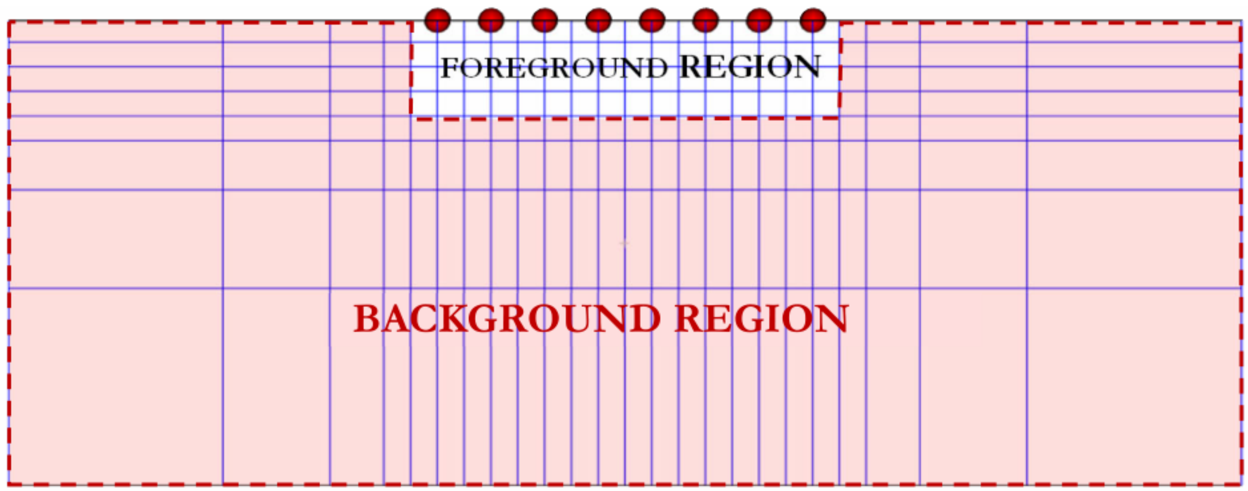


Figure 23.17: In red, the background region

The dimensions of the *Background Region* are defined by the pads: each number manage the position of a background node. The number *n* means *n* times the size of the foreground element size, e.g. if the size of the cell in X is 0.5 m, 1 mean 0.5m (1x0.5), 2 mean 1m (2x0.5), 4 mean 2m (4x0.5) and 8 mean 4m (8x0.5) (Figure 23.18).

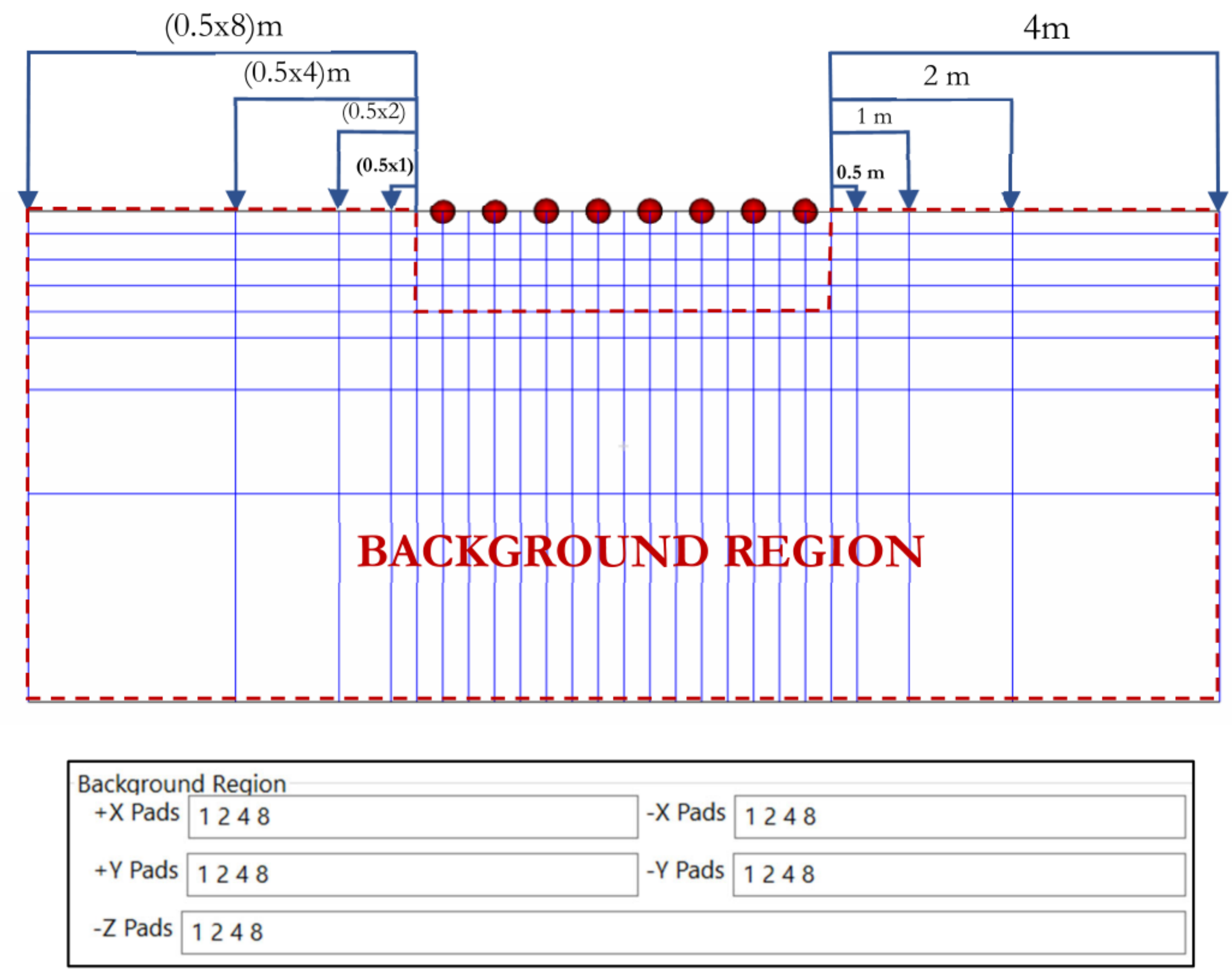


Figure 23.18: Definition of background limits through pads

It is possible to increase or decrease the dimensions of the background by adding or deleting the number of pads in the table. The pad numbers may be different in X, Y and Z direction. Clicking on the *Compute Optimal Value* button will automatically calculate the pads until 64.

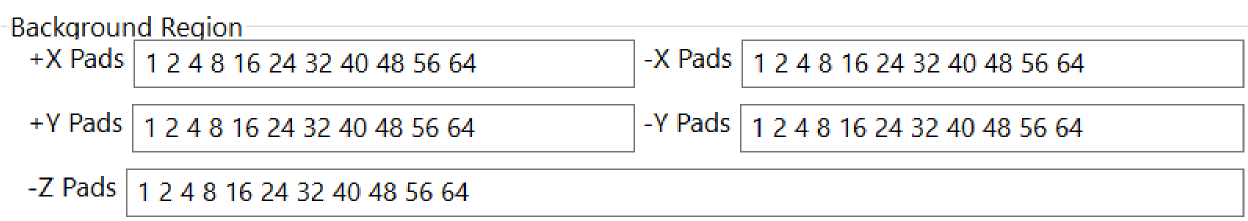


Figure 23.19: Pad setting

If the user changes the pads in the Background region from the automatically calculated pads he needs to make sure that the remote pole (if it exists) is included in the mesh. To do that, after generating the mesh make the remote pole visible (see section Show remote pole) and verify that it is inside the mesh. (unchecking *Foreground Only* tool as explained in section Plot Only Foreground).

If the current settings of the Background Bounds does not include all the electrodes (included the remote poles) a warning is shown, and pressing the button "*Correct*" it is possible to be helped with an automatic adjustment.

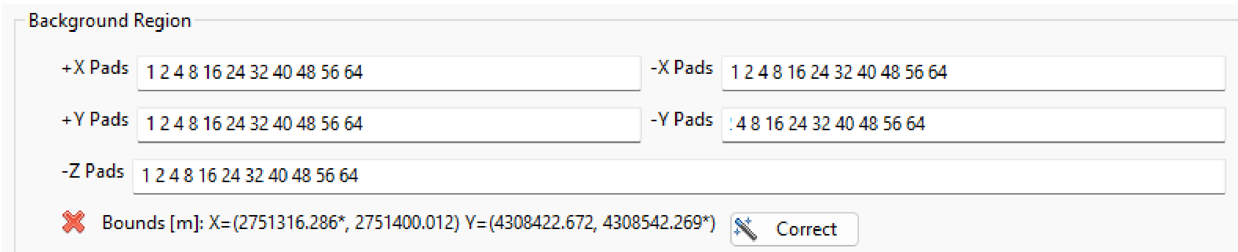


Figure 23.20: Background Bounds Warning

Note that the Background pads depends on the *cell size*, so if it is modified then it is also probably needed to check again the Background Bounds.

23.2 Model

23.2.1 Background value

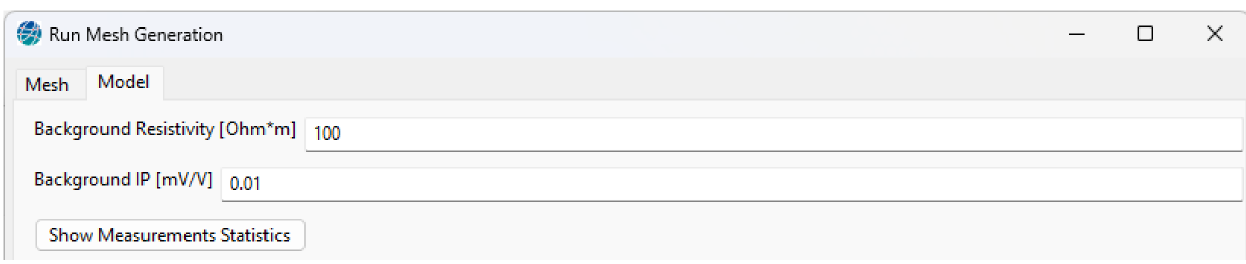


Figure 23.21: Background value

In the *Model* tab it is possible to set the Background Resistivity and IP values that will be used as starting values during the inversion process. The default values are automatically calculated through "*Compute Optimal Value*" function. The user can change them typing the desired values in the appropriate edit boxes. To make the right choice it is useful to consider the median and the average value in the Resistivity/IP Histogram panel (more details in section Histogram).

A quicker way to get the statistics of the measurements is to press the button "*Show Measurements Statistics*".

23.2.2 Anomalies

Moreover, through the table, it is possible to insert one or more anomalies in the Mesh, or a known stratigraphy. Right click on the table and chose "*Append Anomaly*".

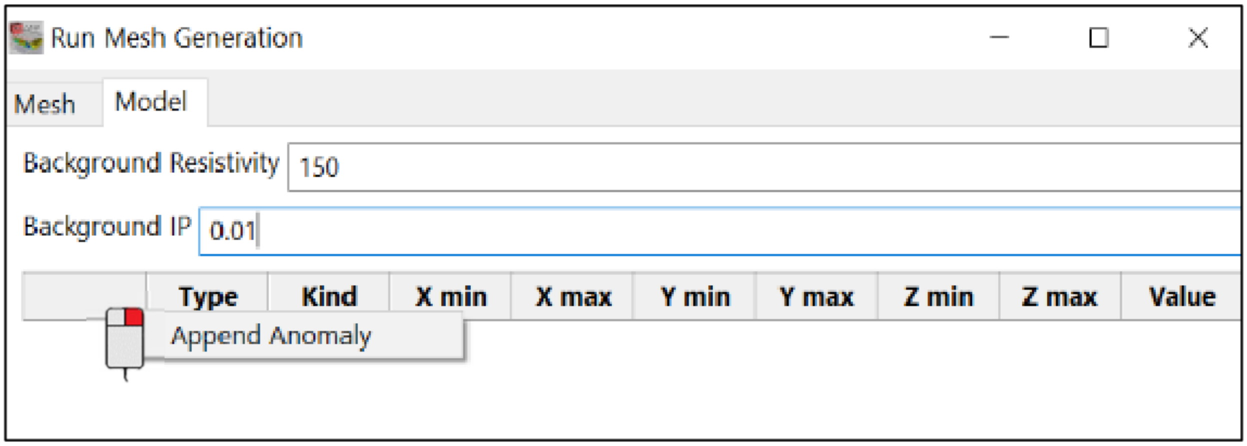
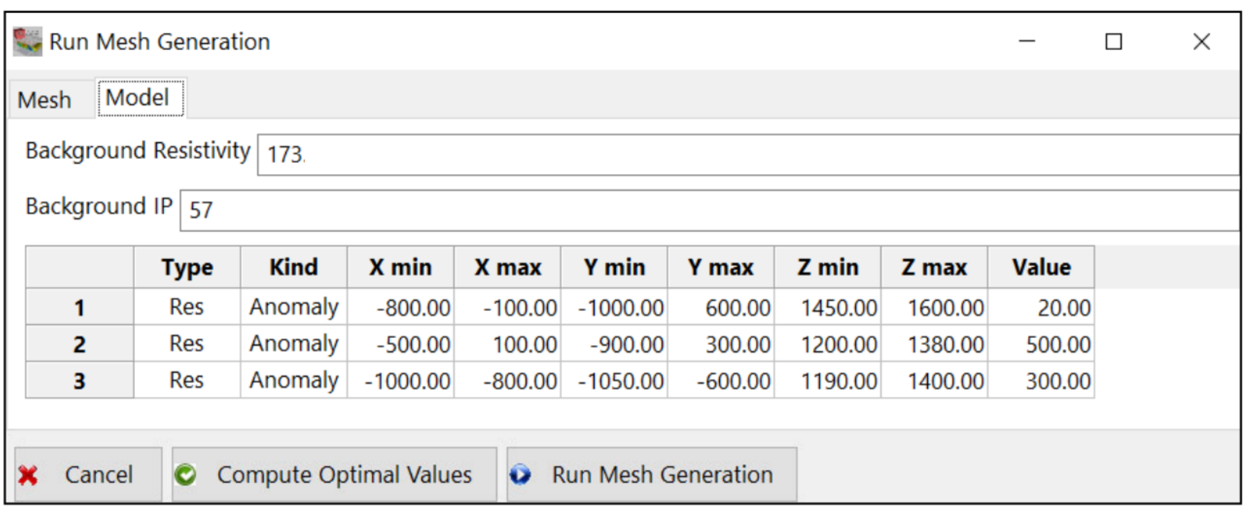


Figure 23.22: Append Anomaly in model table

An empty row will be added to the table to be filled with the desired values. Repeat this operation to add more anomalies. In the following example three Resistivity anomalies are added to the model. It is possible to choose the position and dimension of the anomaly setting the X, Y and Z coordinates. In the following example, the Background Resistivity is 173 Ohm*m and the Resistivity of the anomalies are 20, 500 and 300 Ohm*m.



	Type	Kind	X min	X max	Y min	Y max	Z min	Z max	Value
1	Res	Anomaly	-800.00	-100.00	-1000.00	600.00	1450.00	1600.00	20.00
2	Res	Anomaly	-500.00	100.00	-900.00	300.00	1200.00	1380.00	500.00
3	Res	Anomaly	-1000.00	-800.00	-1050.00	-600.00	1190.00	1400.00	300.00

Figure 23.23: Three customized anomalies added to the Model

The same procedure is applicable for the insertion of IP anomalies, clicking with the right mouse button on the *Type* box and choosing IP instead of Resistivity.

To remove an anomaly (of any type) right click on the specific raw of the table and choose "*Delete Anomaly"*.

To actually take the anomalies into account during the inversion, check the "constrain to reference model" button in the inversion panel (see section Inversion Tab).

Clicking on "*Run Mesh Generation*" will create the mesh, once the **Mesh** and the **Model** tabs have been filled with the appropriate parameters. This process can take some processing time depending on the mesh size and the number of cells.

To customize the Mesh visualization mode use the dedicated node in section Mesh and Model. The variations of Resistivity values of the model can be displayed in several ways. Through contour lines, volumes, surfaces or sections (all these options are explained in detail in section Mesh and Model) (Figure 23.24).

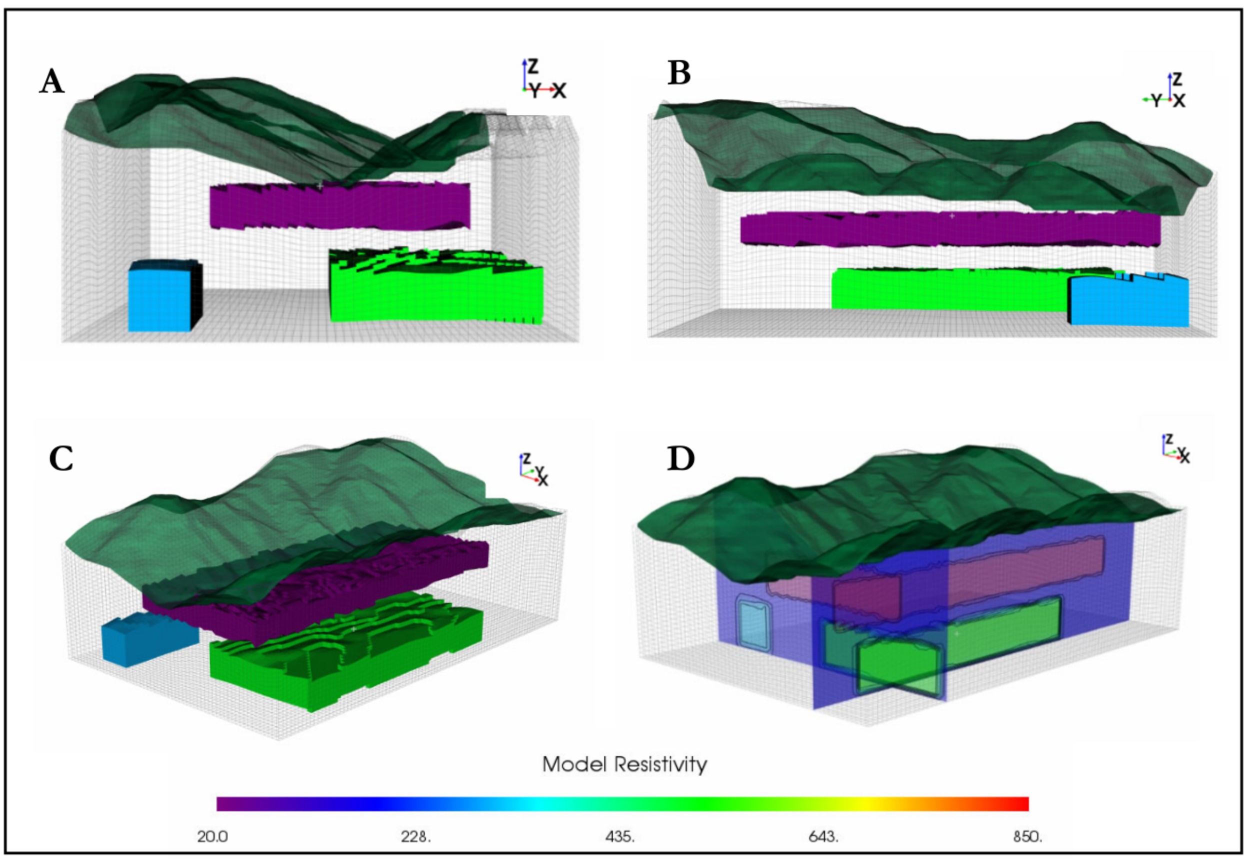


Figure 23.24: Example of different way to display the starting model

In the A, B, and C cases the topography (in dark green), the mesh (grey cells) and the three model anomalies (in purple, light green and light blue) are displayed as volumes, from two different points of view. In the case D the topography (in dark green), the mesh (grey cells) and the three anomalies are displayed through two vertical sections.

CHAPTER TWENTYFOUR

RUN FORWARD MODEL

By clicking with the right mouse button on the main node of the "*ERT Data*" and selecting "*Run Forward Model*" opens a new window for the forward modelling.

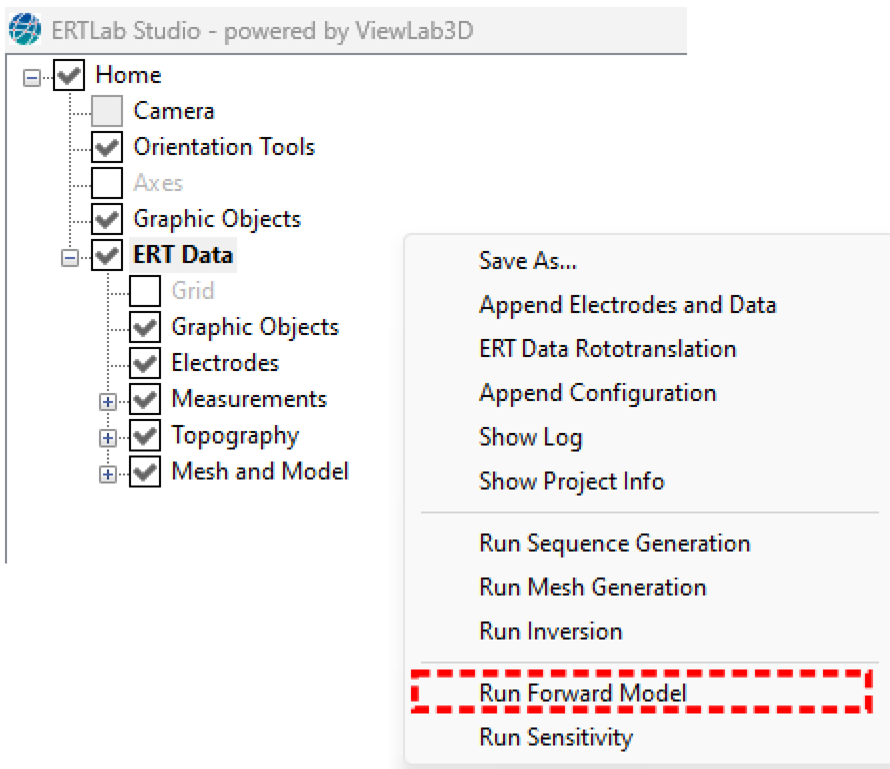


Figure 24.1: Forward Model

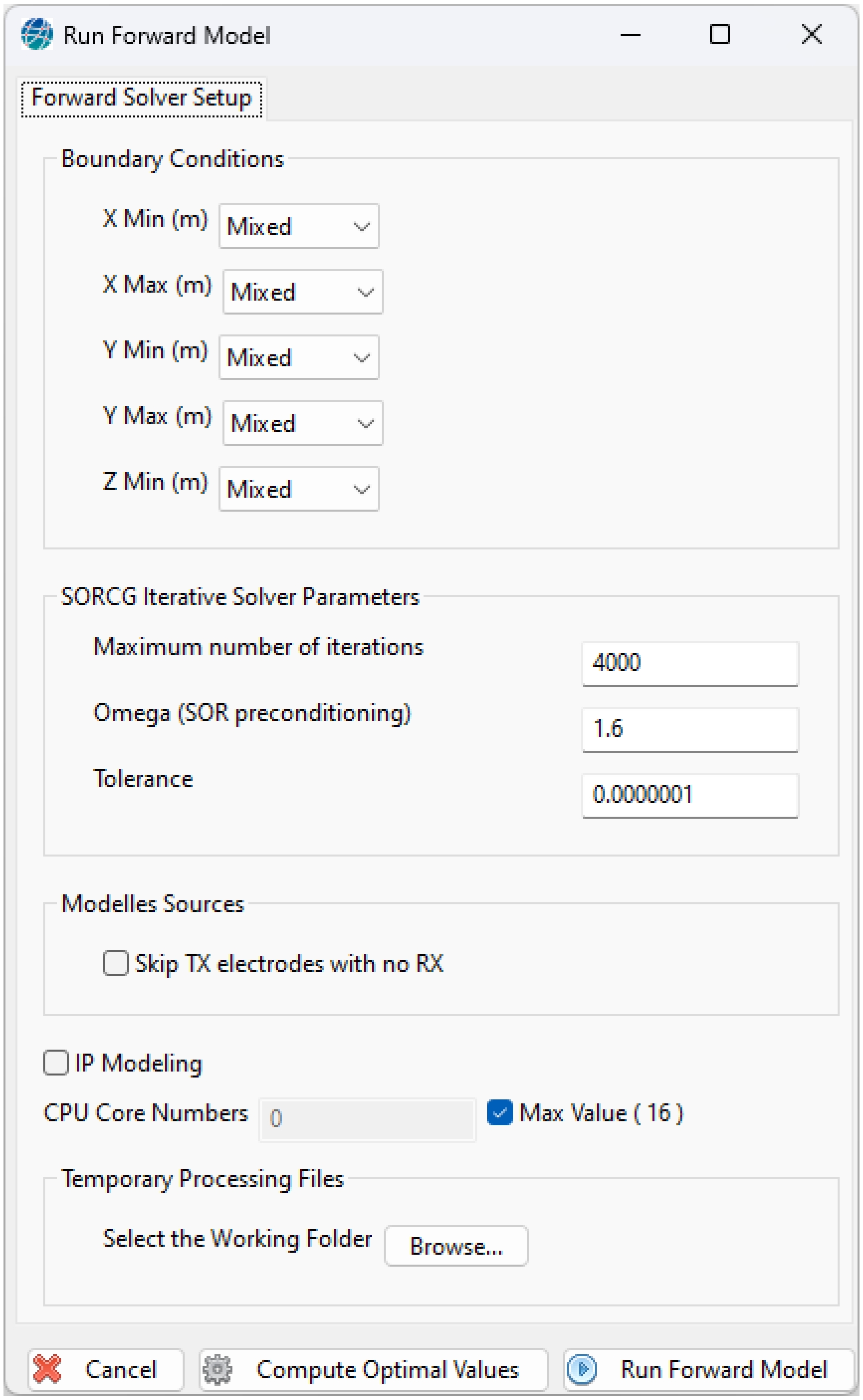


Figure 24.2: Forward Model Panel

In this window the user can set the boundary conditions and the solver parameter for the forwards model. To define the area under investigation, the half-space domain is delimited with artificial boundaries that simulate planes in horizontal and vertical direction as infinitely distant. The values for these borders "infinitely distant" (-X, +X, -Y, +Y, -Z, +Z) can be specified using the dedicated selection boxes:

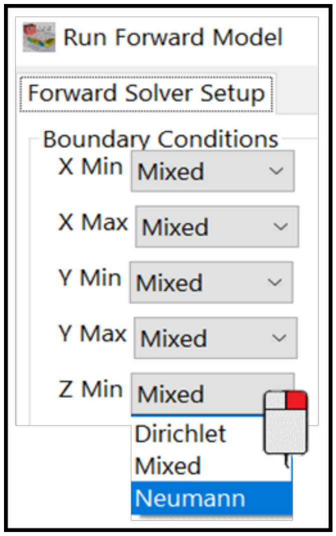


Figure 24.3: Choice of boundary conditions

- **Dirichlet condition**, in which the potential is known and set equal to zero: V=0. This is the typical condition of an "infinite" half-space in which the potential decreases to zero as the distance from the electrodes increases.
- **Neumann condition**, in which the current flow is zero (derivative of the potential equal to zero): ∂ V / ∂ η =0, with η equal to z for the bottom edge or equal to x for the lateral ones. In particular, this condition occurs at the air-ground interface, because the conductivity of the air is almost zero and in all those cases in which the half space is abruptly interrupted in one direction (vertical walls, big topographical jumps, etc.)
- **Mixed Condition,** the Dirichlet condition may lead to values of the potential being underestimated (at a certain distance from the sources), while the Neumann Condition may lead to an overestimation.

In this case it can be useful to use a Mixed boundary condition. It the case of having a remote pole, which is typically close to the outer edges of the mesh, the potential is still not zero at the boundary.

By default, the boundaries of all directions are set to Mixed condition.

In the same panel it is possible to set other parameters as shown in Figure 24.4

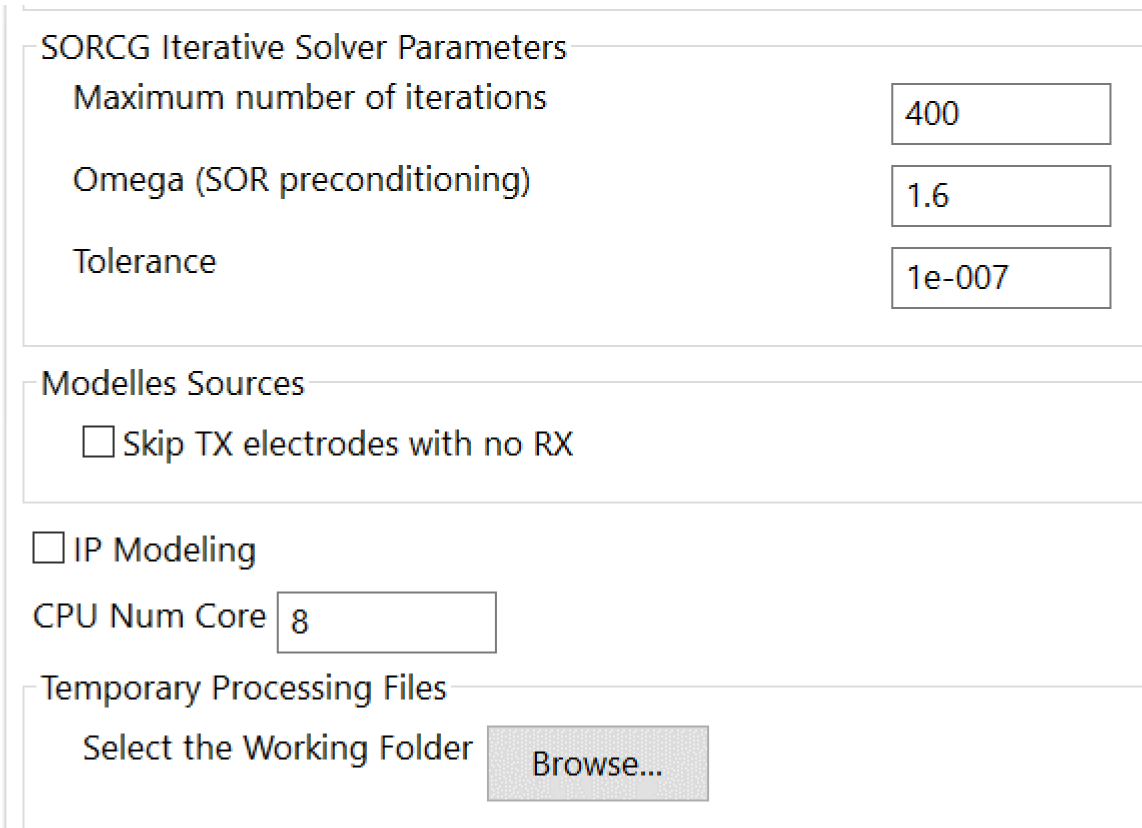


Figure 24.4: Forward solver setup panel

SORCG Iterative Solver Parameters:

- **Maximum Number of iterations**: when the number of conjugate gradient iterations exceeds this value, the computation stops: the forward modelling solution does not get the convergence within the requested tolerance for the modelled electrode.
- **Omega (SQR preconditioning)**: this is the value of the conjugate gradient preconditioner in the forward modelling solution. Users can ignore this parameter.
- **Tolerance**: this is the value of tolerance requested in the iterative conjugate gradient forward modelling solution.

Modelled Source:

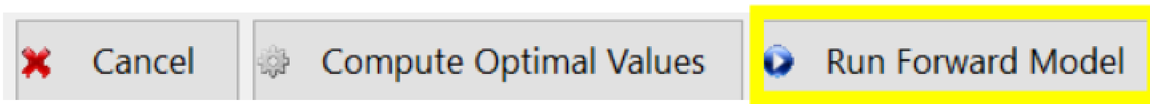
- **Skip TX electrodes with no RX**: by checking this command the TX electrodes will be skipped during calculation in the finite elements modelling to get faster runs during forward modelling-only tasks.

Other Parameters:

- **IP Modelling**: with ERTLab *Studio* it is possible to invert Electrical Resistivity (Rho) and Induced Polarization (IP) models at the same time. To include the IP data in the inversion process, check this IP box.

- **CPU Core Numbers**: this value depends by the computer hardware features on which you are working. The more threads can be used for the inversion the faster the processing will be running.
- **Temporary Processing Files**: it lets to choose where to save the folder with the temporary files of the intermediate steps of the inversion process.

When all the parameters are manually set or automatically calculated by pressing the *Compute Optimal Values* button, click on *Run Forward Model* to start the process.



CHAPTER

TWENTYFIVE

RUN SENSITIVITY

By clicking with the right mouse button on the main node of the "*ERT Data*" and selecting "*Run Sensitivity*" will open the window shown in Figure 25.2.

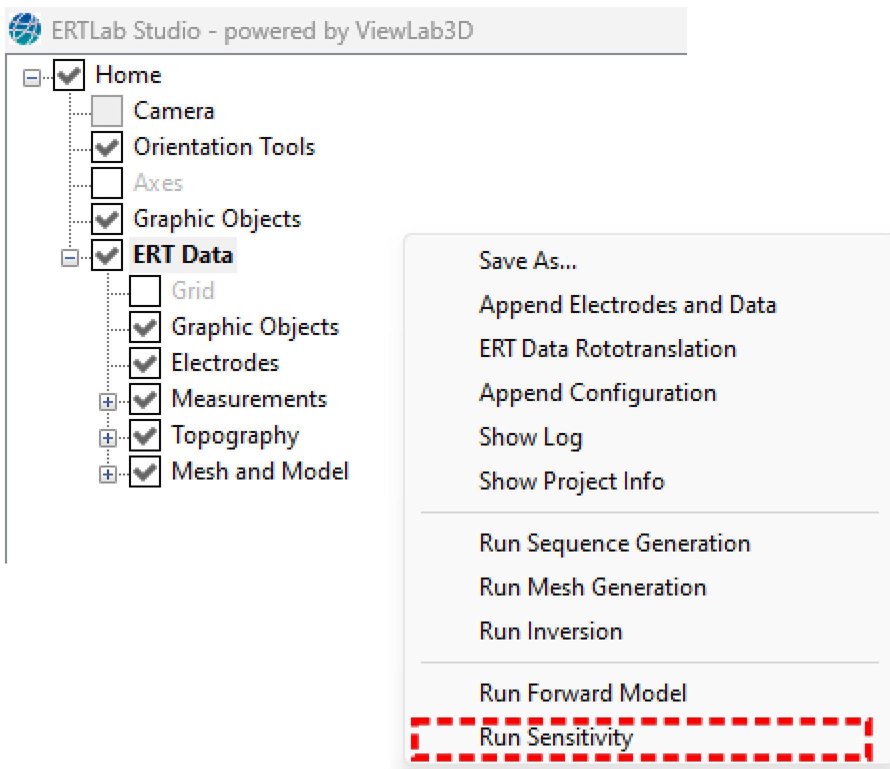


Figure 25.1: Run Sensitivity

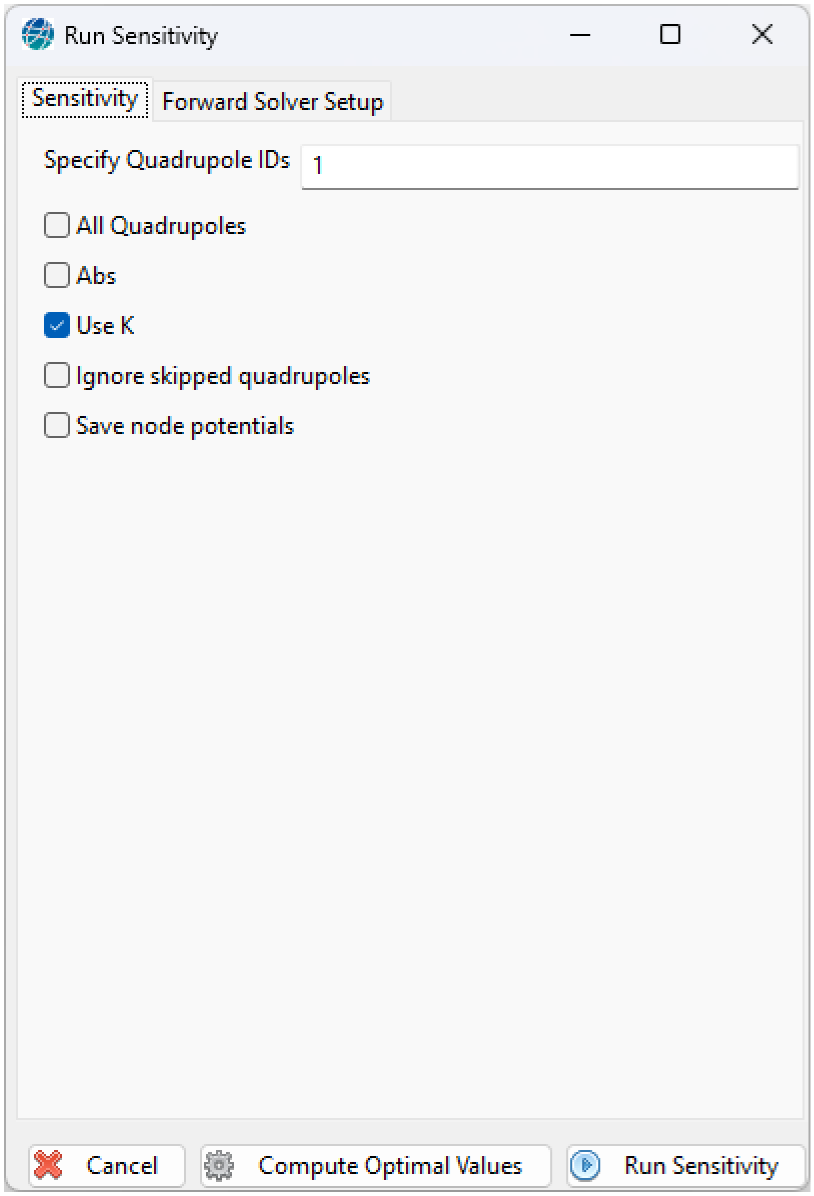


Figure 25.2: Run Sensitivity Panel

The window has two tabs: *Sensitivity* and *Forward Solver Setup*. The latter is already explained in the section Run Forward Model. In the *Sensitivity* panels it is possible to set the number of quadrupoles on which to compute the sensitivity. The sensitivity is an index of how much a change in resistivity of the soil affects the measurement of the potential as the two parameters are directly proportional. The user can compute the sensitivity of a specific quadrupole or of all quadrupoles together.

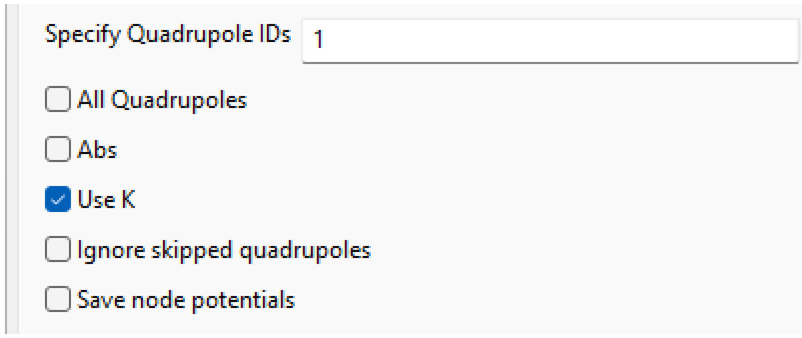


Figure 25.3: Sensitivity panel

In the first case, uncheck the "*All Quadrupoles*" box and type the number of the quadrupoles in the appropriate number box (1 in Figure 25.3).

The *Abs* check box lets the user choose to consider the positive and negative values, or only the absolute values. If more than one quadrupole is specified then the *Abs* mode is automatically enabled and it is not possible to uncheck it to avoid errors in the sensitivity. This because the sum of positive and negative superimposed areas, resulting from different quadrupoles, can lead to uncorrected values of the final sensitivity.

The sensitivity can be computer in term of V/I or in term of the Apparent Resistivity, so set "*Use K*" disabled in the first case, and enabled in the second case.

The sensitivity can be computed on all the quadrupoles, or only on the filtered list, so use the flag "*Ignore skipped quadrupoles*" to select the desired configuration.

For advanced needs, it is also possible to enable "*Save node potentials*" to let be saved in the temporary folder used during the later computation a text file with the potential for each node of the mesh.

Clicking on the *Run Sensitivity* button will start the process. A progress message and a *Process Completed* message will appear during and at the end of the calculation (Figure 25.4).

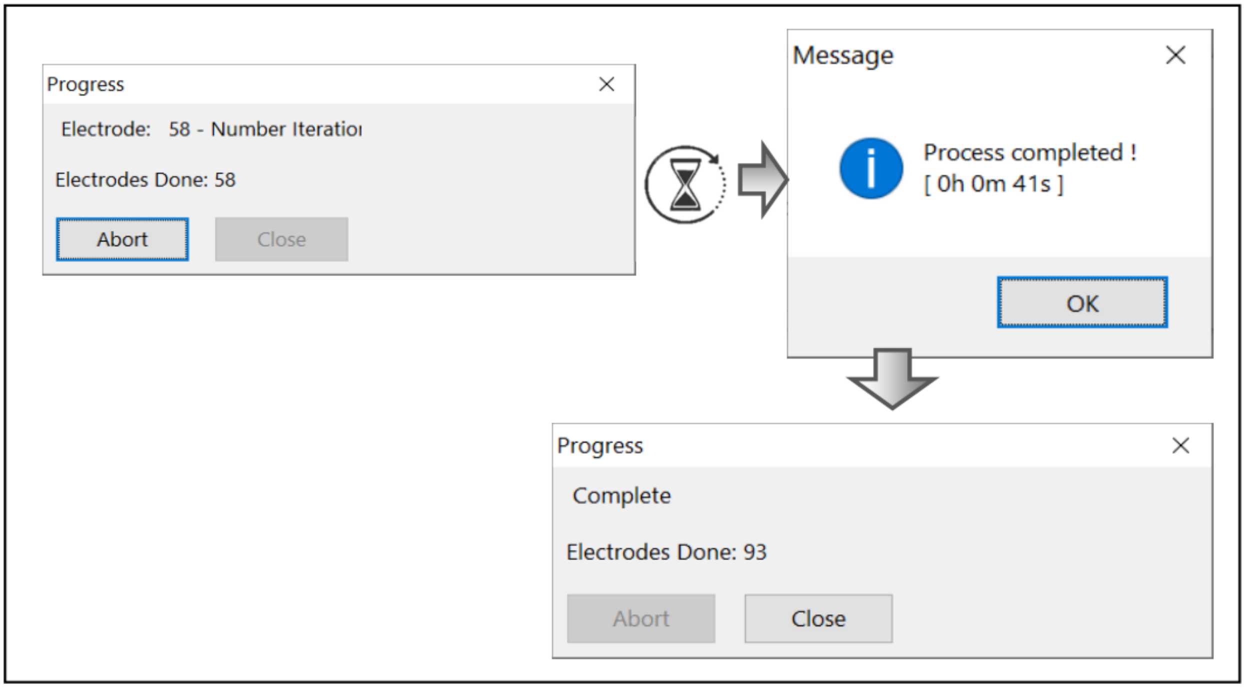


Figure 25.4: Sensitivity Model computation

The results can be visualized using the "*Sensitivity Model*" option (sub-node of Mesh and Model option, section Sensitivity Model). In the example in Figure 25.5, a full quadrupoles sensitivity is calculated and visualized trough sections and volumes. As expected, the maximum sensitivity is in proximity of the electrodes.

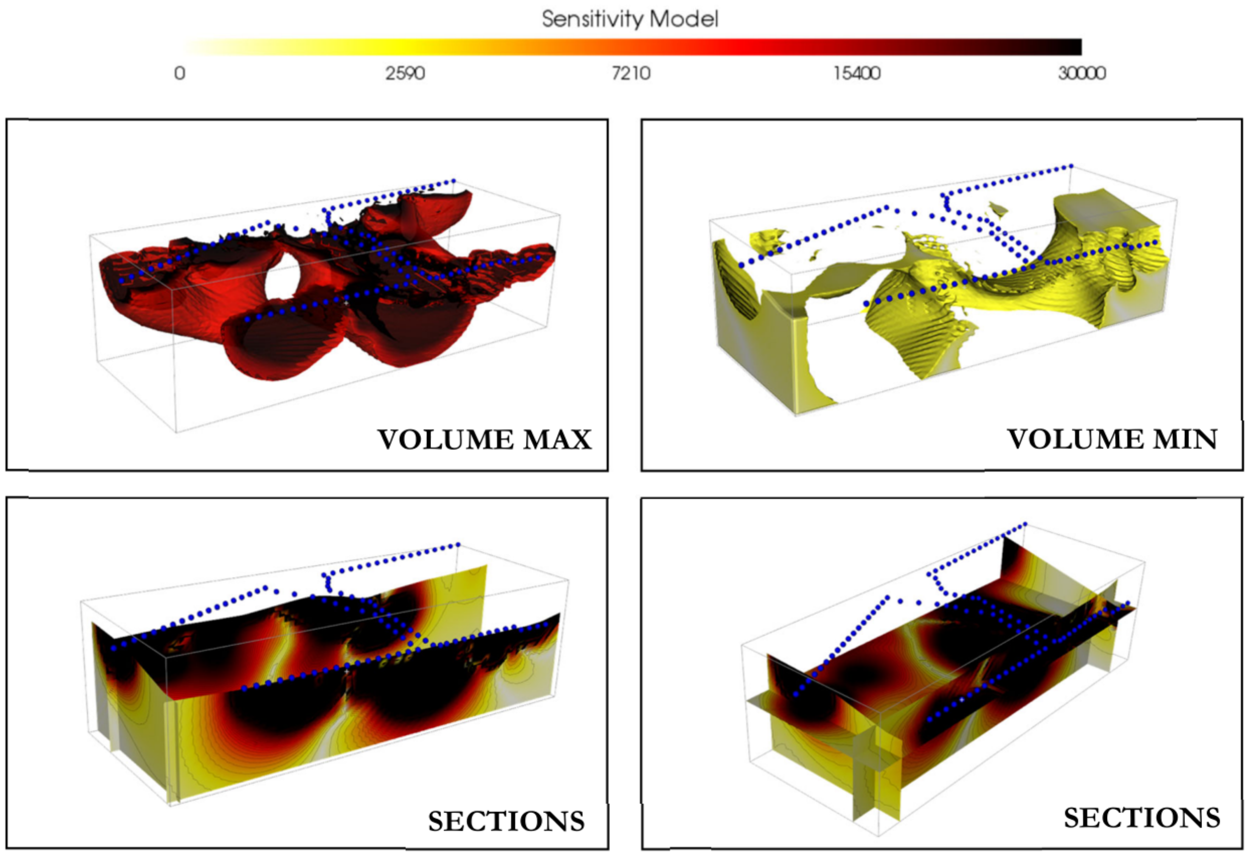


Figure 25.5: Example of different visualizations of Sensitivity Model

CHAPTER

TWENTYSIX

RUN INVERSION

By clicking with the right mouse button on the main node of the "*ERT Data*" and selecting "*Run Inversion*" will open the window shown in Figure 26.2.

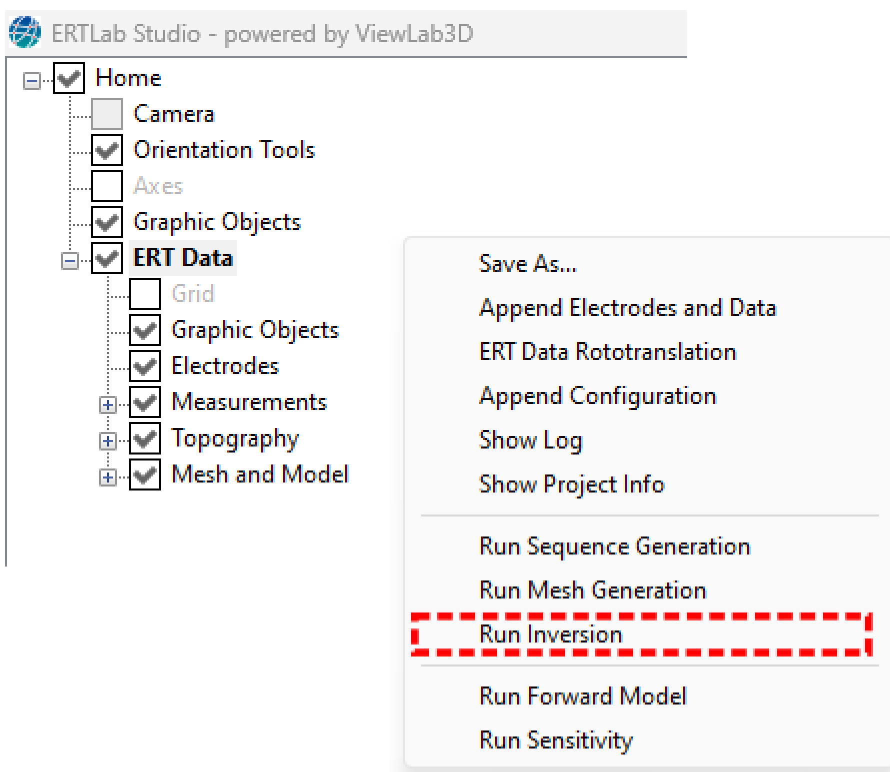


Figure 26.1: Run Inversion

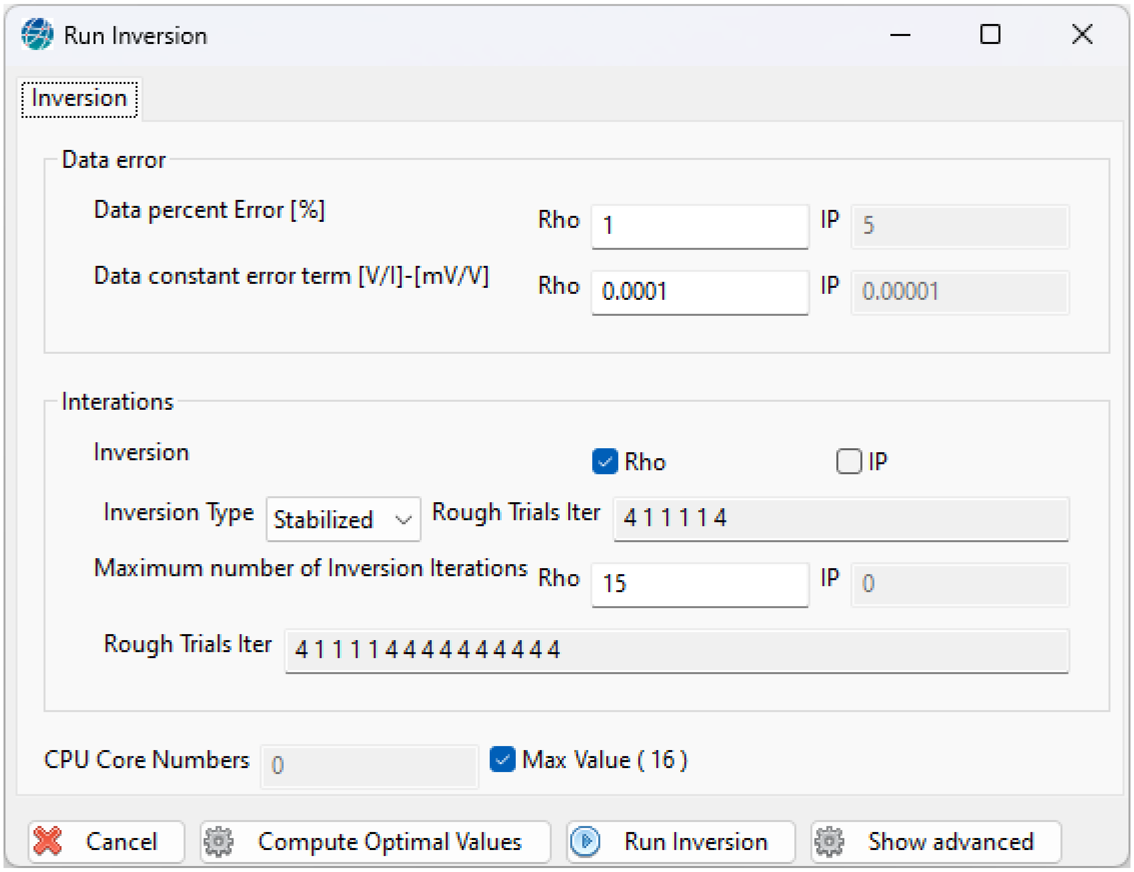


Figure 26.2: Run Inversion Panel

26.1 Simple view

26.1.1 Data Error

In a good model the difference (*misfit*) between the acquired data and modelled data is sufficiently small (it solves the inversion problem). However, field data are affected by errors (*noise*). For this reason, the inversion processing does not search for the models that exactly reproduce the field data, but for the model which reproduces the filed data without a certain noise. In this window all the mathematical parameters to estimate this error can be set before running the inversion. In this window the user can set the Error of the Rho and IP in terms of percent error or constant value (suggestions are: 1 for very clean data, 3-5 for data with medium signal/noise ratio, 10 or more for very noisy data).

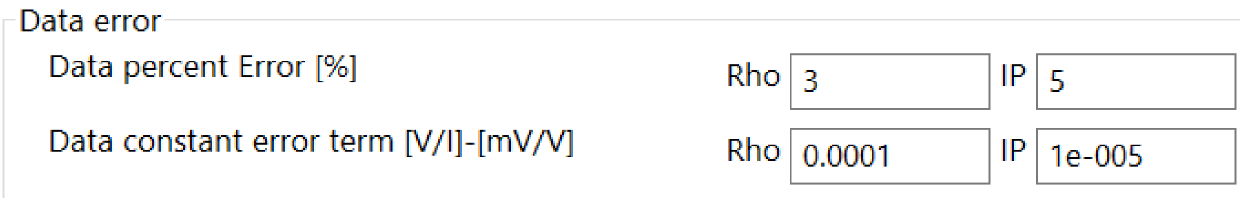


Figure 26.3: Data error panel

- Data Percent Errors [%]: This parameter controls the acceptable percent difference between the data and the model (the inversion will converge to this error level). The value entered here should reflect the estimated degree of data repeatability (i.e. the actual noise in the field data), which is easily obtained if reciprocal data were collected. This term works in conjunction with the constant error term to determine convergence.
- Data Constant Error Term [V/I]-[mV/V]: This parameter indicates the acceptable absolute difference between the data and the model that is acceptable (the inversion will converge to this error level). The value entered here should reflect the estimated lower noise threshold of the instrument used to collect the data. This term works in conjunction with the percent error term to determine convergence.

26.1.2 IP Modelling

ERTLab Studio can simultaneously process Electrical Resistivity (Rho) and Induced Polarization (IP) data. To include the **IP** data in the inversion, check the box behind *IP*.

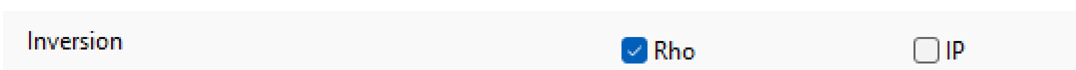


Figure 26.4: IP Modelling panel

26.1.3 Iterations

The inversion process computes "trials", to determine the optimal roughness parameters to use each iteration. This operation can take a long time, so the user can choose the number of trials to run on each iteration. Left clicking on the *Inversion Type* drop down box exposes three possible selections:

- *Simple* (4 1): it performs 4 trials at the first iteration and 1 trial from the second iteration onwards.
- *Full* (4): it performs 4 trials at each iteration, from the first to the last.
- *Balanced* (4 1 1 1 4 1 1 1 …): it performs 4 trials only at specified iterations, else only 1 trial will be performed.
- *Stabilized* (4 1 1 1 1 4): it is similar to the *Simple* configuration for the first 5 trials, then it proceeds with 4 trials up to the end.

- *Custom*: it lets the user choose the number of trials at each iteration, writing the desired numbers in the dedicated box.

In *Maximum number of Inversion Iterations* the user can set the number of iterations to compute the Rho and the IP separately. The complete sequence of the trials iterations resulting from the set values is shown in the *Rough Trials Iter* box (Figure 26.5). In the example a custom inversion is shown, where a maximum of 15 iterations are to be performed. At the first iteration 4 trials are performed, 2 at the second, and 1 from the third to the last iteration (maximum 15).

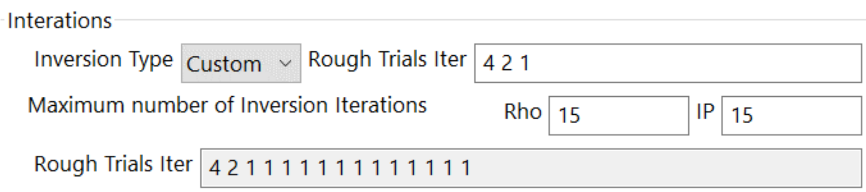


Figure 26.5: Custom iteration

26.1.4 CPU Core Numbers

This value depends on the hardware features of the computer used to run ERTLab Studio. The more threads can be used for the inversion the faster the processing will be running. An estimation of the maximum number of thread can make automatically enabling the "*Max Value*" flag.

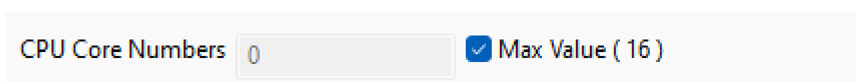


Figure 26.6: CPU setting

26.2 Advanced view

Clicking on the *Show Advanced* button will expose additional options (Figure 26.7).

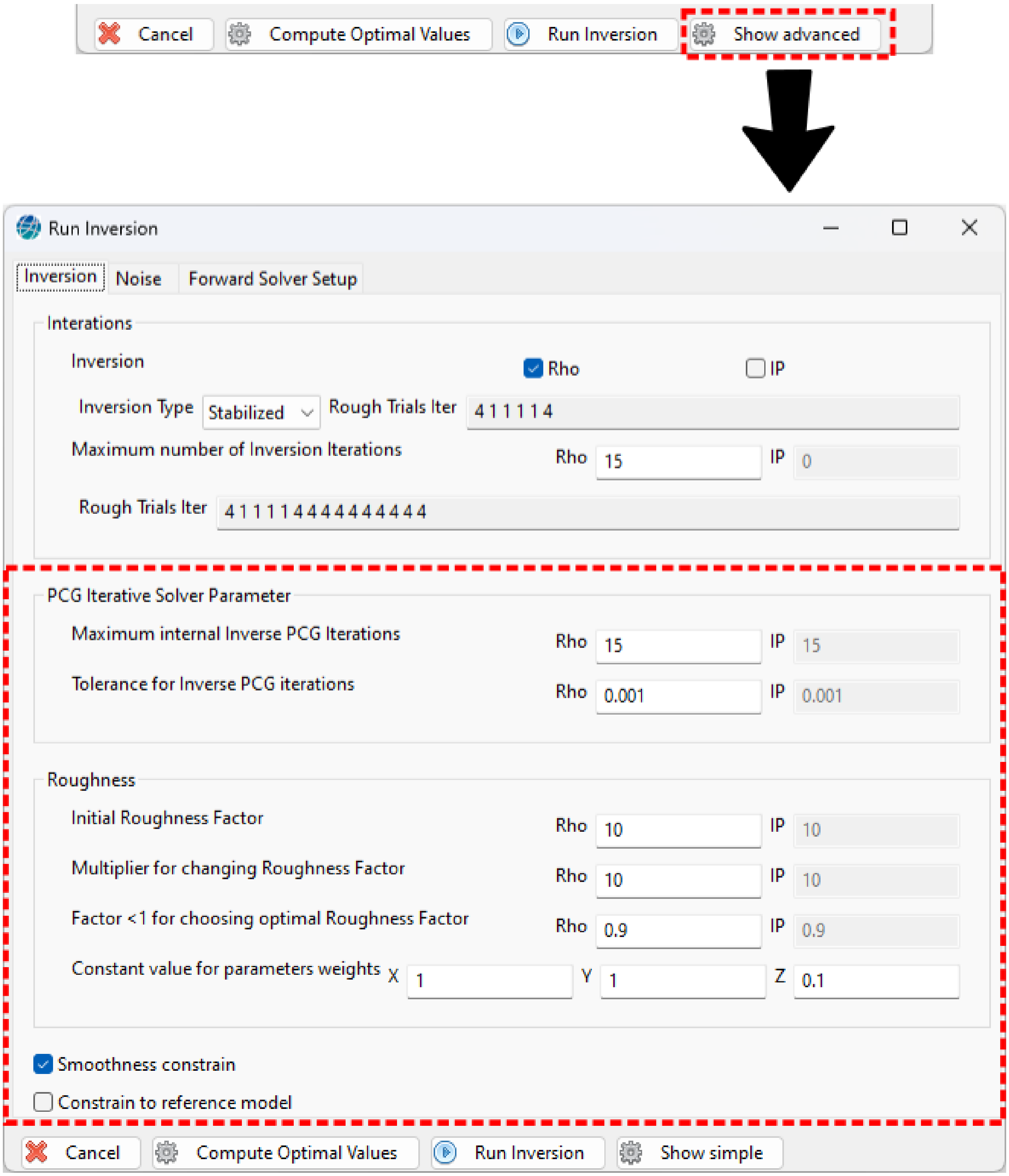


Figure 26.7: Advanced inversion panel

The window has three tabs: *Inversion*, *Noise* and *Forward Solver Setup*. Some options are the same as already explained in this User Guide. In addition, it is possible to set the following parameters.

26.2.1 Inversion Tab

PCG Iterative Solver Parameters:

- **Maximum internal Inverse PCG Iterations**: this is the maximum number of "inner" iterations in the conjugate gradient inversion solver at each iteration. The higher the number of iterations the more the solution of the system of equations will approach the analytical solution, but the slower the inversion. A value of 15 is usually enough. For simple 2D inversions this value can be increased to 30.
- **Tolerance for Inverse PCG iterations:** this is the requested tolerance for the conjugate gradient inversion solver.

Roughness parameters:

- **Initial Roughness Factor**: lets the user set the value to use as *Roughness Factor* at the beginning of inversion process. Almost always the default (10) is the best choice.
- **Multiplier for changing Roughness Factor**: when it the simple inversion type is used (trials 4 1), the roughness parameter uses results from the previous roughness divided by the multiplication factor set through this tool.
- **Factor < 1 for choosing optimal Roughness Factor:** the optimal roughness factor is chosen based on the minimum data residual with a given tolerance, which is defined by this factor. Typically, users can ignore this parameter.
- **Constant value for parameters weight**: these weights of the roughness parameters allow to control the roughness trend in the three directions, x, y and z. By default, they are the same in the three directions.

Other parameters:

- **Smoothness Constrain**: when selected, the smoothness constrain of adjacent cells will be applied to both model blocks variations and model blocks values. If not selected the smoothness constrain of adjacent cells will only be applied at the model blocks variations "dm_i" at each iteration "i".
- **Constrain to reference model**: selecting this command will constrain the solution to the starting model at each iteration. If during the model setting one or more anomalies have been inserted (section Anomalies). Users have to check this box to take them into account during the inversion.

26.2.2 Noise Tab

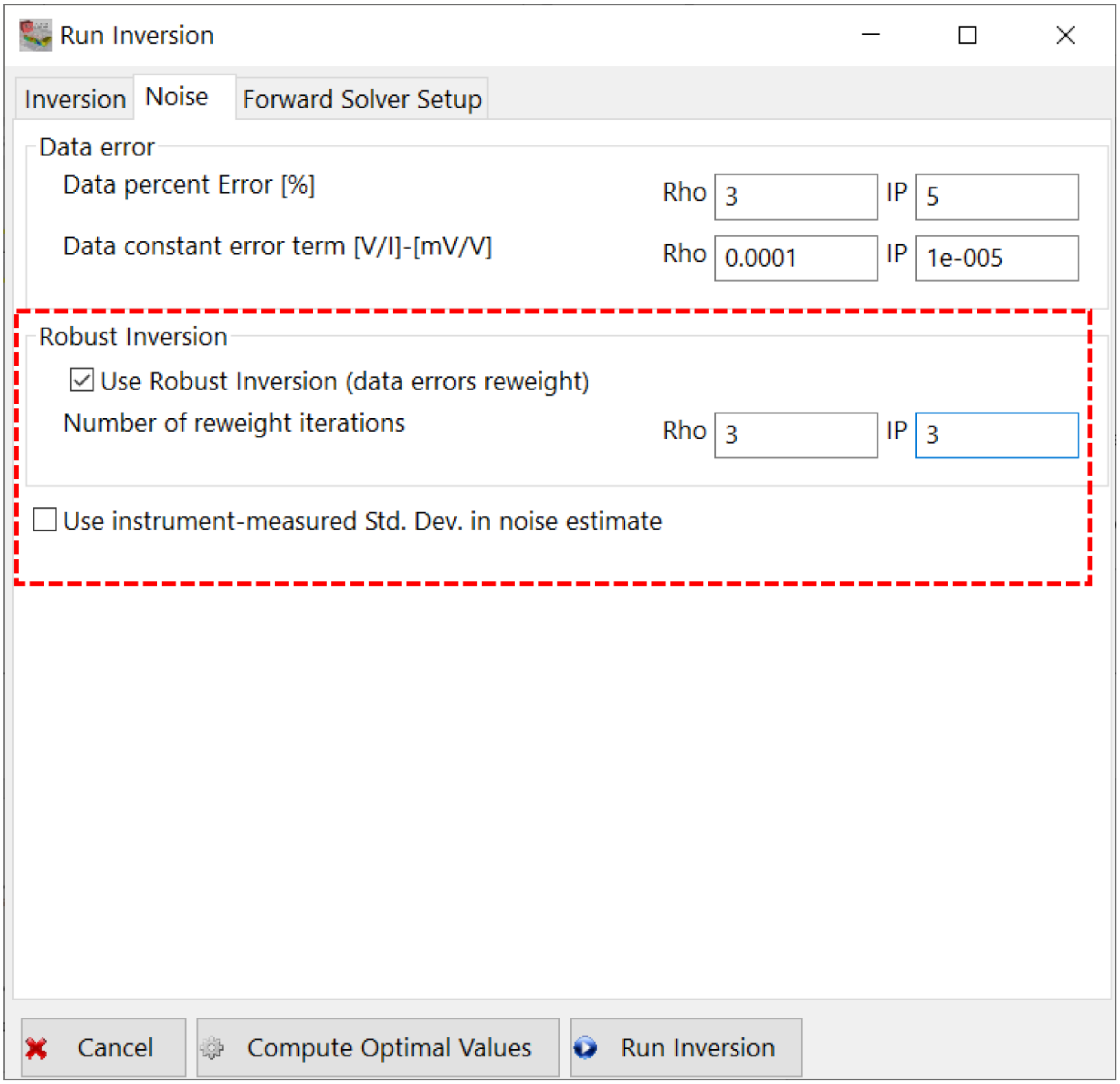


Figure 26.8: Advanced noise panel

- **Use Robust inversion (data errors reweighted):** generally, the model does not constitute exactly the real distribution of data but just an approximation of it, so it is advisable to adopt a Robust Inversion. In this case, the results are relatively insensitive to changes in assumptions of the statistical model. At the contrary, in case of not robust inversion there will be abrupt variations even in correspondence to small changes of data distribution. If "*Use Robust Inversion*" is selected, the noise is appropriately modified during the inversion after each iteration, to reduce the "weight" to those measures which are heavily affected by errors (outliers) and therefore are not relevant to the model.

It is possible to set even the number to reweight iteration for Rho and for IP separately.

26.2.3 Forward Solver Setup Tab

This tab is the same as already explained in section Run Forward Model.

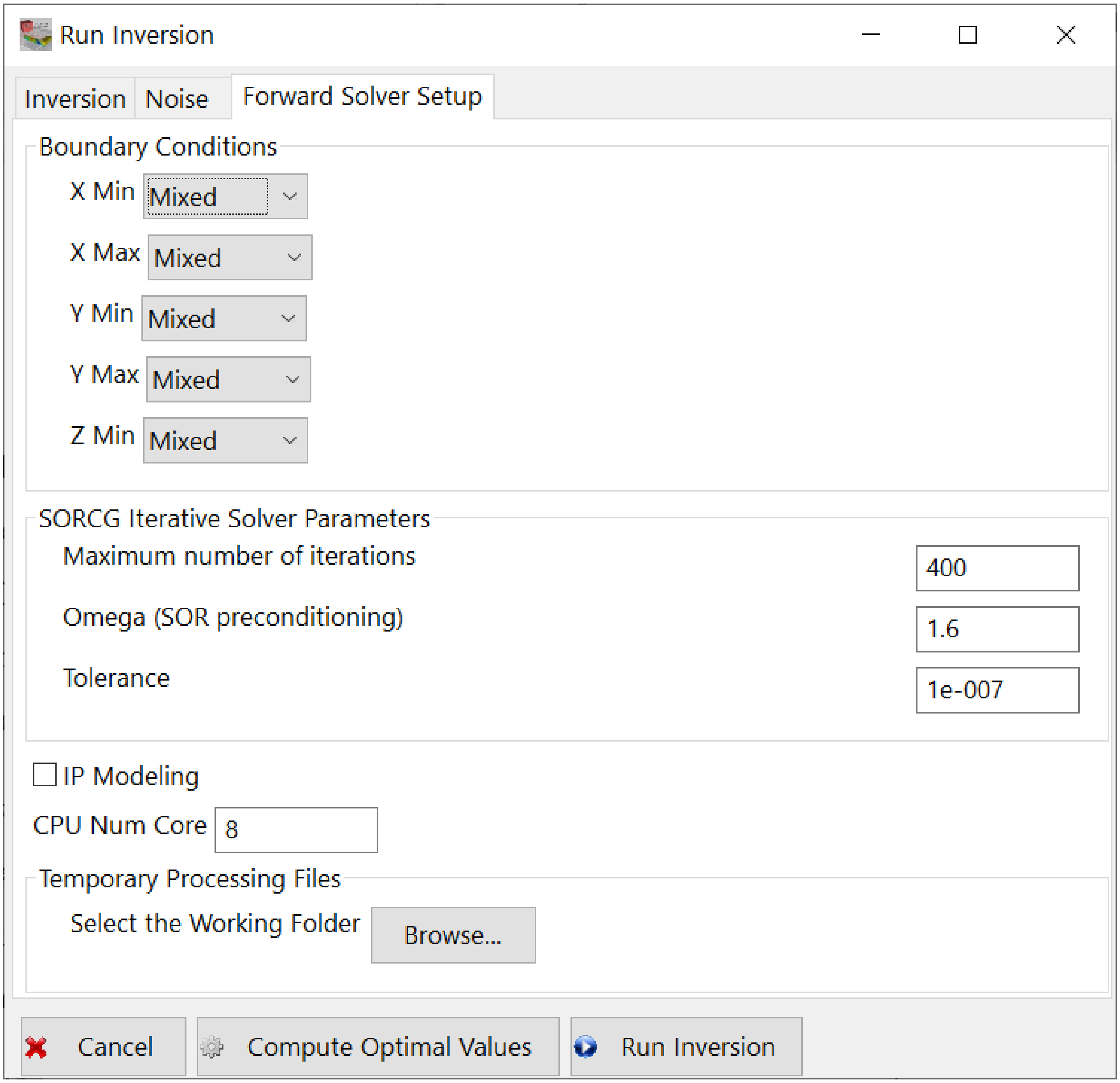


Figure 26.9: Forward Solver Setup panel

26.3 Progress Window

When all the parameters in all three tabs are set, click on the *Run Inversion* button to start the inversion. Chose and select a project folder where the inversion files will be automatically saved. A progress window will appear on the screen and it will be completed automatically as the calculation proceeds. When the processing ends, a message will alert the user (Figure 26.10).

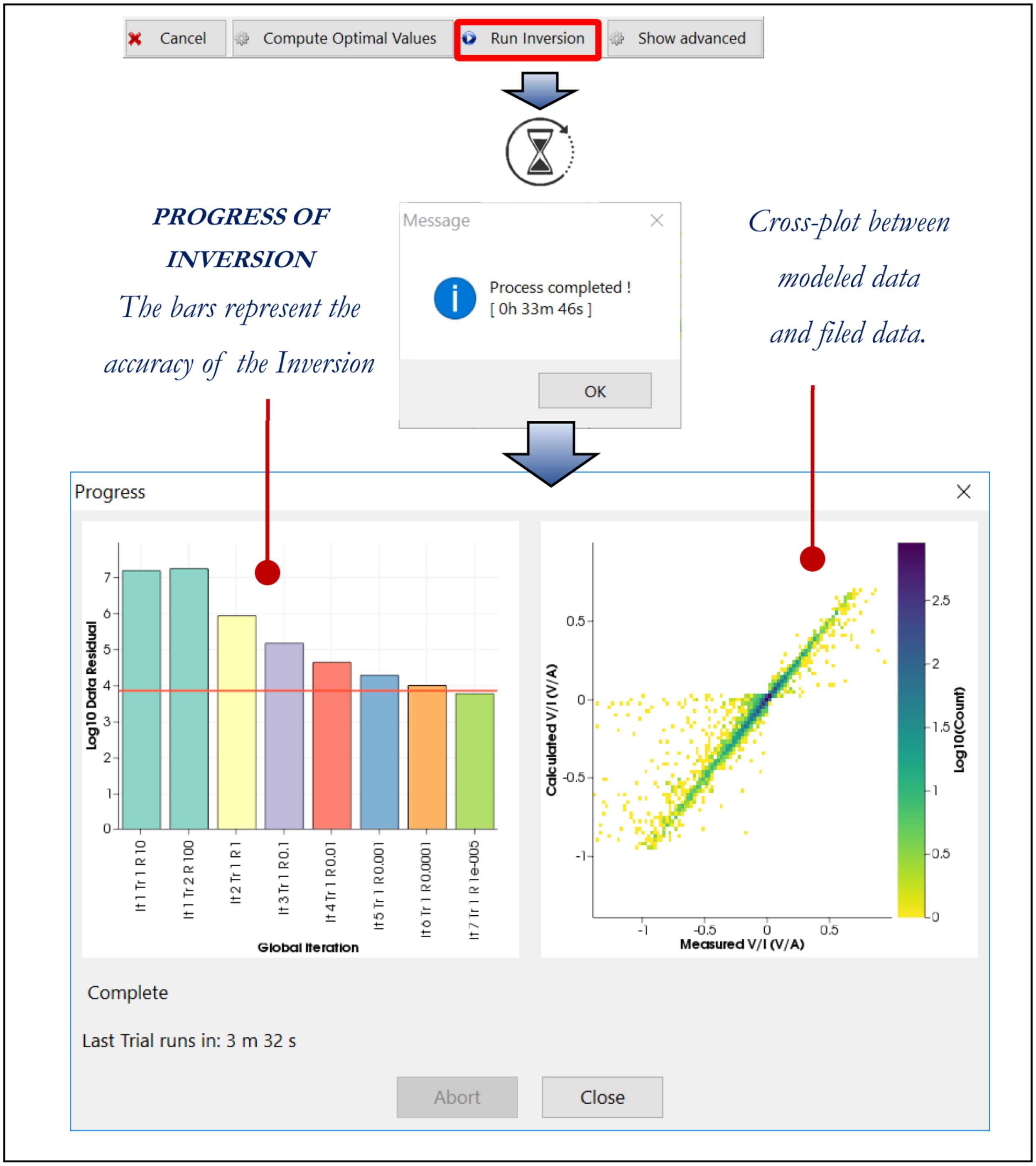


Figure 26.10: Run Inversion progress

The inversion summary chart is automatically saved in the project folder (chosen when the inversion was started) with the name “RES Iter **n** Trial **m**.png” (where *n* is the number of iteration and *m* the number of trial). A good inversion gives a diagram where:

- The bar height decreases as iterations proceed (as a consequence of the residual decreasing);

- The last bar of the bar chart corresponds in height to the red line (Data residual, ideal inversion target = number of measurements to process).
- There is a low number (7 in the example) of iterations (easy convergence).

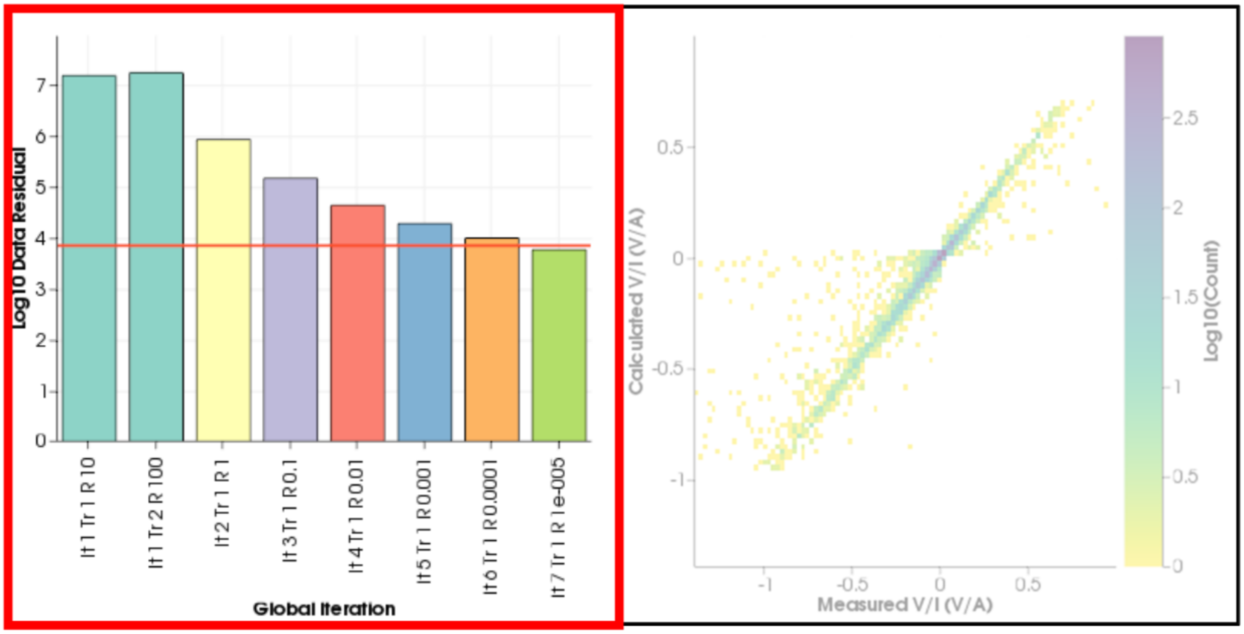


Figure 26.11: Histogram chart of a reliable inversion

- At the end of the inversion, the plot between field data and calculated data is near to 1:1 ratio, and data are distributed along the diagonal;
- Abnormal values (outliers, yellow dots, where the absolute difference between modelled data and measured data is high) are in the minority.

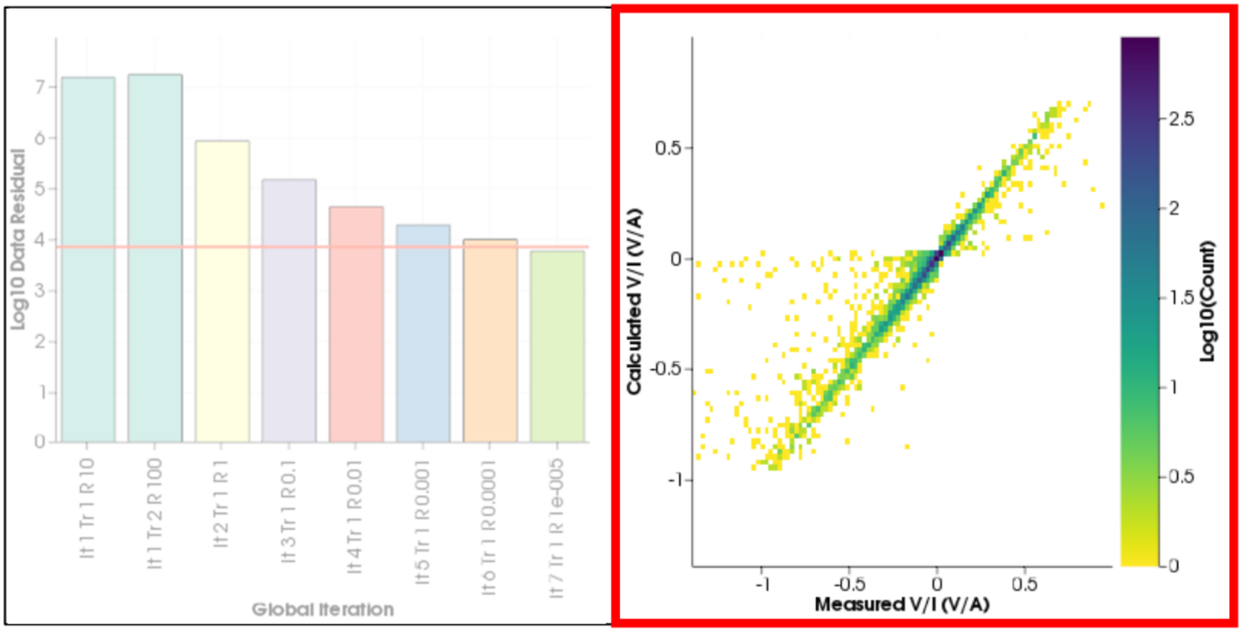


Figure 26.12: Cross-plot of a reliable inversion

Figure 26.13 shows an example summary chart of a *not* successful inversion.

- The histograms remain at a constant height from iteration 6 to 12 (no progresses at the proceed of inversion) and at the last 2 iterations they are opposite to the trend, reaching very high residual values;
- The last bar in the bar chart does not match the red target line;
- High number of iterations (14 iterations) (difficulty in convergence).

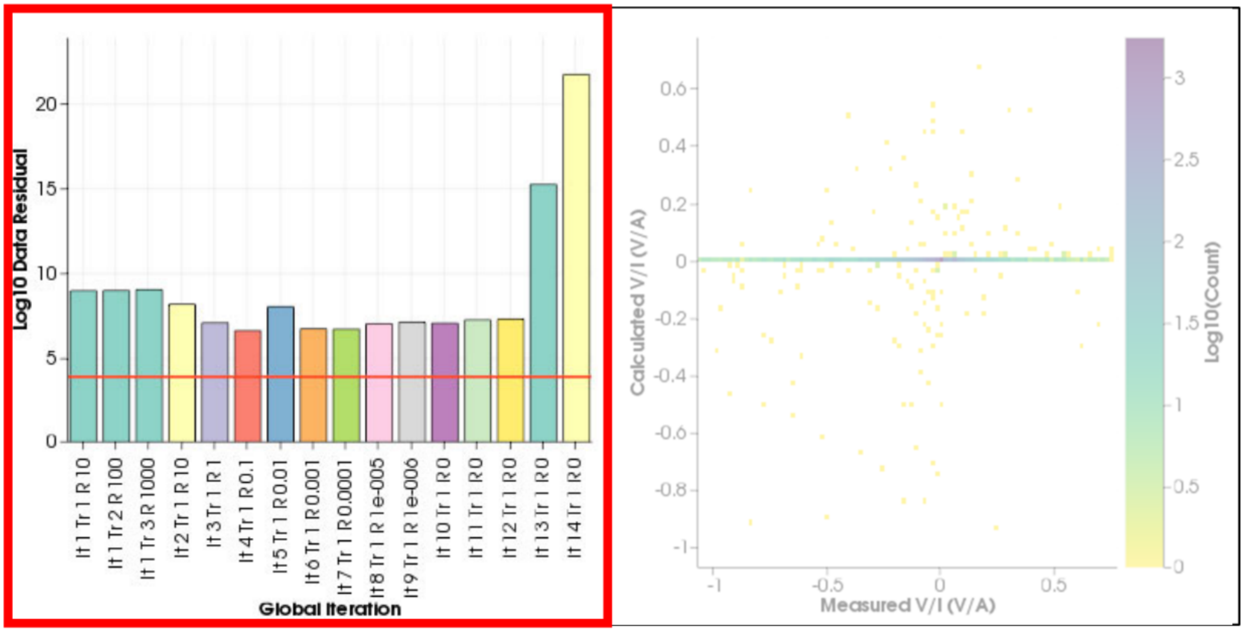


Figure 26.13: Histogram chart of a not reliable inversion

- At the end of the inversion, the plot between field data and calculated data is far from the diagonal (in this case they are align along 0 of calculated V/I);
- Outliers (yellow dots, where the absolute difference between modelled data and measured data is high) are the most common values and they are distributed almost evenly throughout the cross-plot.

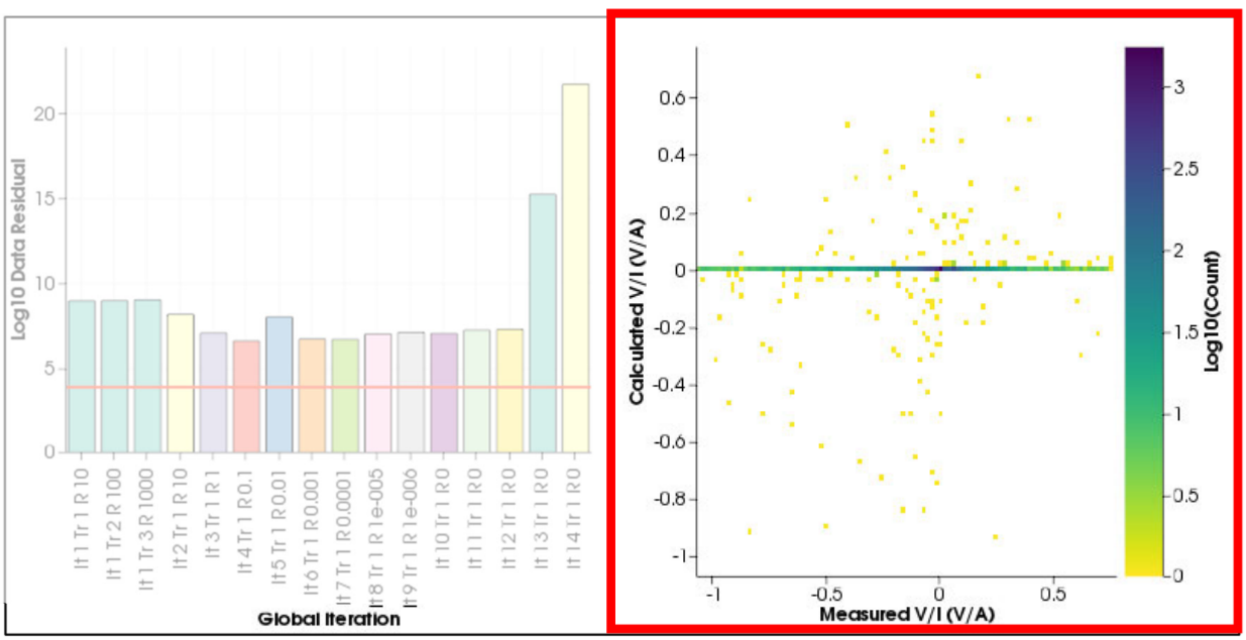


Figure 26.14: Cross-plot of a not reliable inversion

In this case it is necessary to check and eventually further clean the field data, edit the noise, and/or the starting model before proceeding to run the inversion again.

CHAPTER

TWENTYSEVEN

ROTOTRANSLATION

To set the Position, the Rotation, and the Scale of the object (useful if there is the necessity to emphasize one dimension). Some examples are shown in Figure 27.1. This is only a *"graphical" rototranslation*, the coordinates of the datafile do not change. It is a useful tool to visualize the project in a coordinates reference system different from the current one, in which the data were collected (in this case the axes numbers will refer to the new coordinates, but when exporting the datafile nothing has changed in it and it still is in the original coordinate system). It is also useful to make some quick evaluation to find the correct values to apply to the ERT data, to actually rototranslate them through the proper tool (see "*ERT Data Rototranslation*" command on ERT Data).

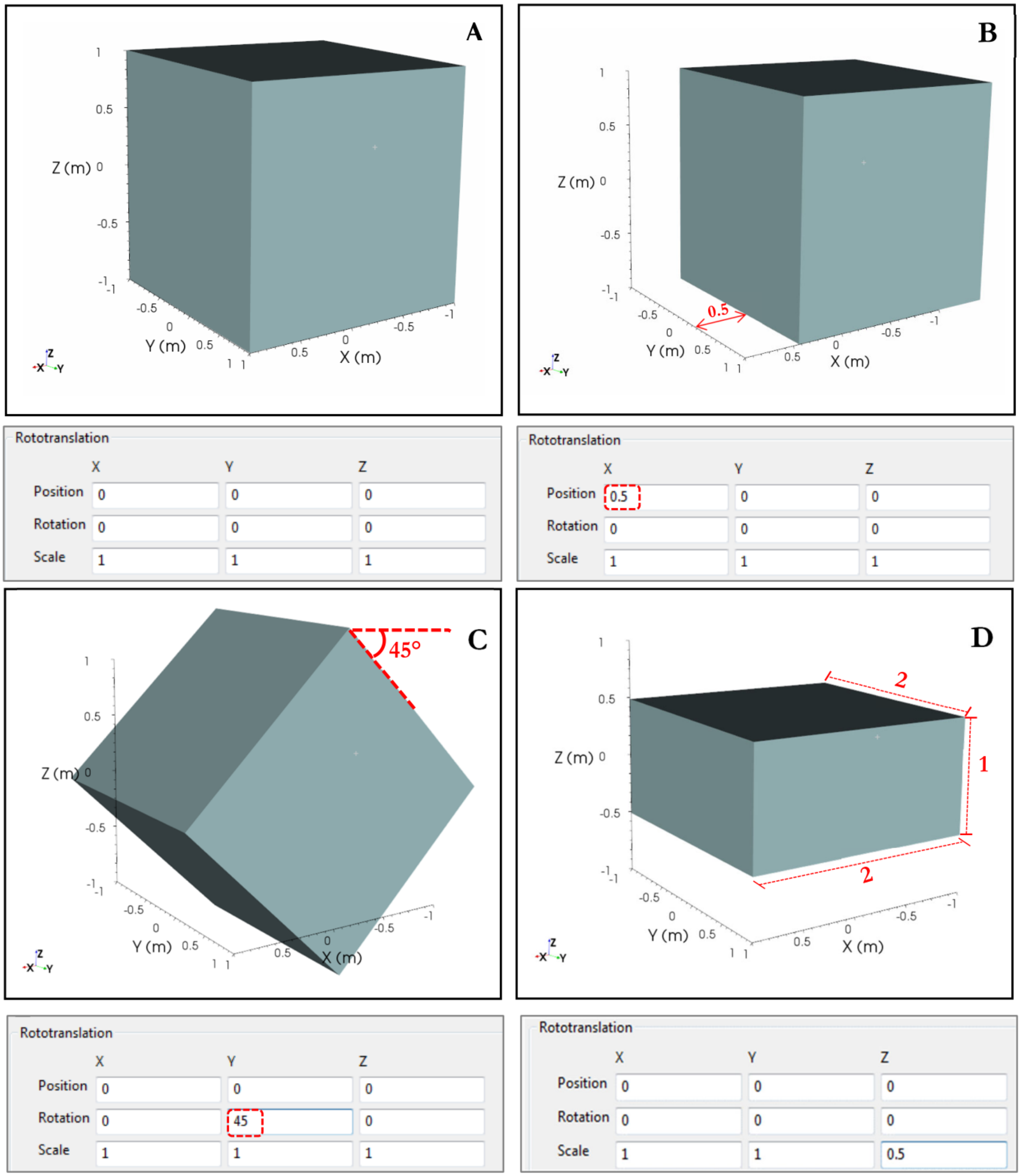


Figure 27.1: Some example of rototranslation. In A the cube in the original position, with the default values in the table; in B translation along X axis of 0.5; in C rotation along Y axis of 45° and in D decrease in the scale ratio of 0.5 along Z axis

Through the Position X, Y, and Z it is possible to choose the position of the object in the scene; through the Rotation X, Y, and Z it is possible to rotate the object up to 360° in the three directions, and through the Scale X, Y, and Z, it is possible to change the scale of the entire 3D scene (sometimes it may be useful to change the scale of one direction to emphasize the trend of one property, for example an alignment along one direction).

Using the button "Last Picked" it can be easily set the X,Y,Z Position from a mouse click (see Picker setting).

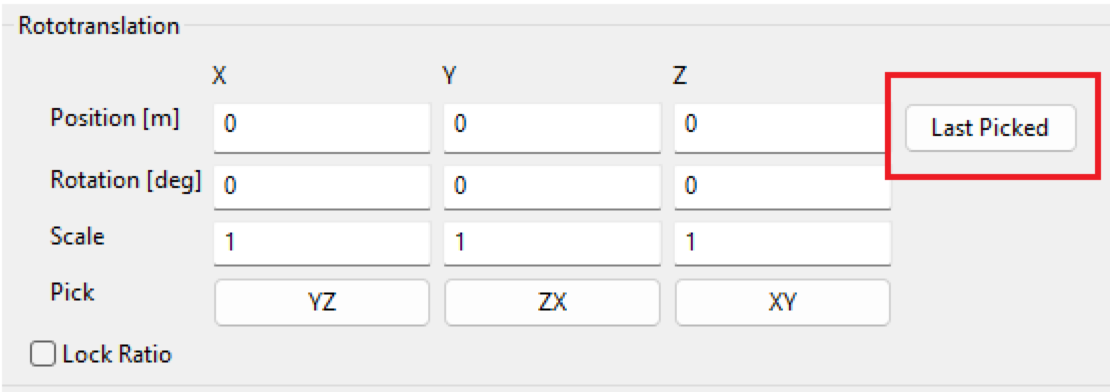


Figure 27.2: Last Picked point button

This table is useful when it is necessary to place the scene to a specific viewpoint and/or rotate it by a defined quantity, expressed by an exact number. Otherwise, if it is required to manually shift the scene, it is possible to use the dedicated **Pick buttons**.

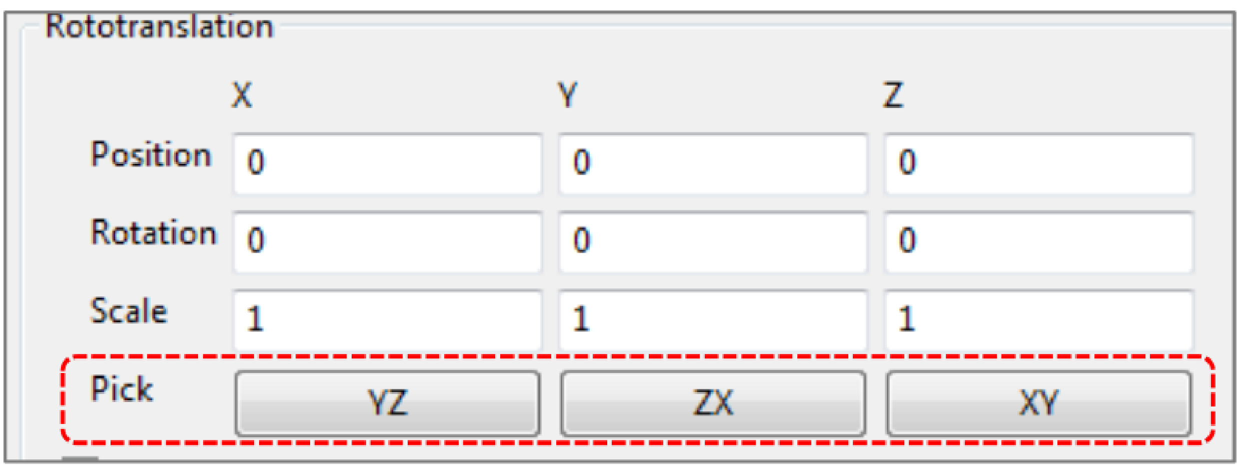


Figure 27.3: Pick Button in rototranslation panel

Clicking on one of these buttons, it is possible to act along the selected plane. A specific panel will appear overlaid to the object, as it is shown in Figure 27.4.

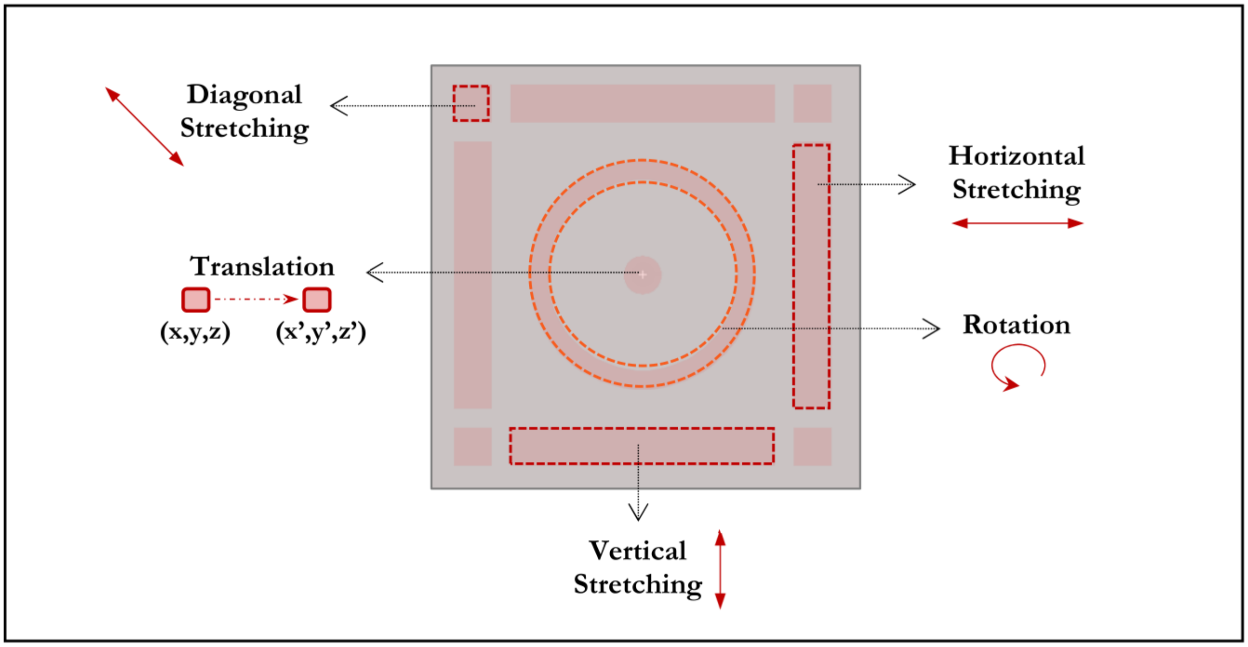


Figure 27.4: Manually rototranslation panel

For example, to stretch one cube along X axis drag the mouse until the desired position while holding the right mouse button. The X dimension will be changed (from 1 to 2 in this example) and the other two dimensions will remain unchanged (Figure 27.5).

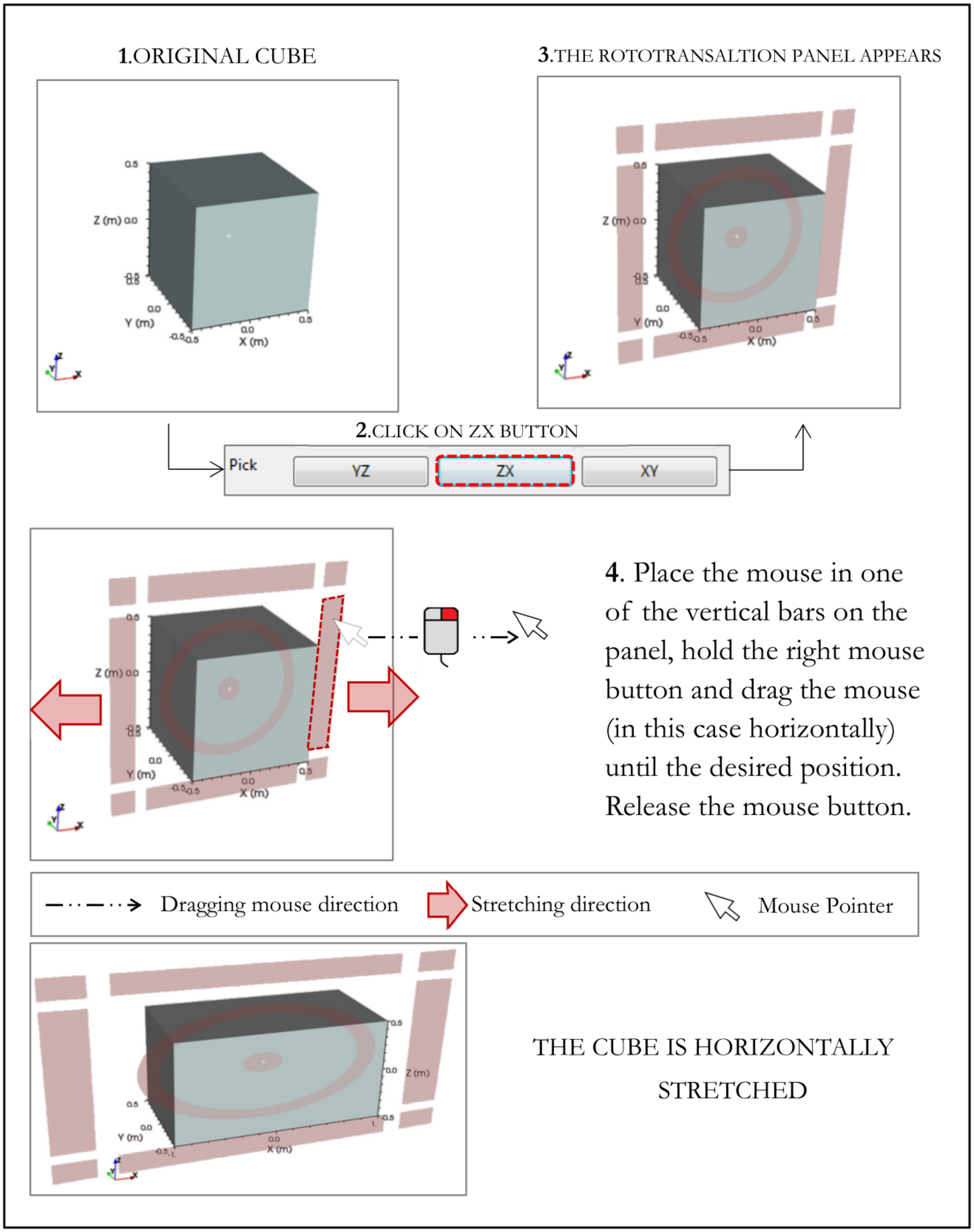


Figure 27.5: Example of manually stretching of a cube in one direction

If **Lock Ratio** is checked, dragging the mouse in one direction even the others two dimensions of the same plane will change of the same relative quantity. In the following example, the cube is deformed vertically through the mouse, but it is stretch horizontally too (Figure 27.6).

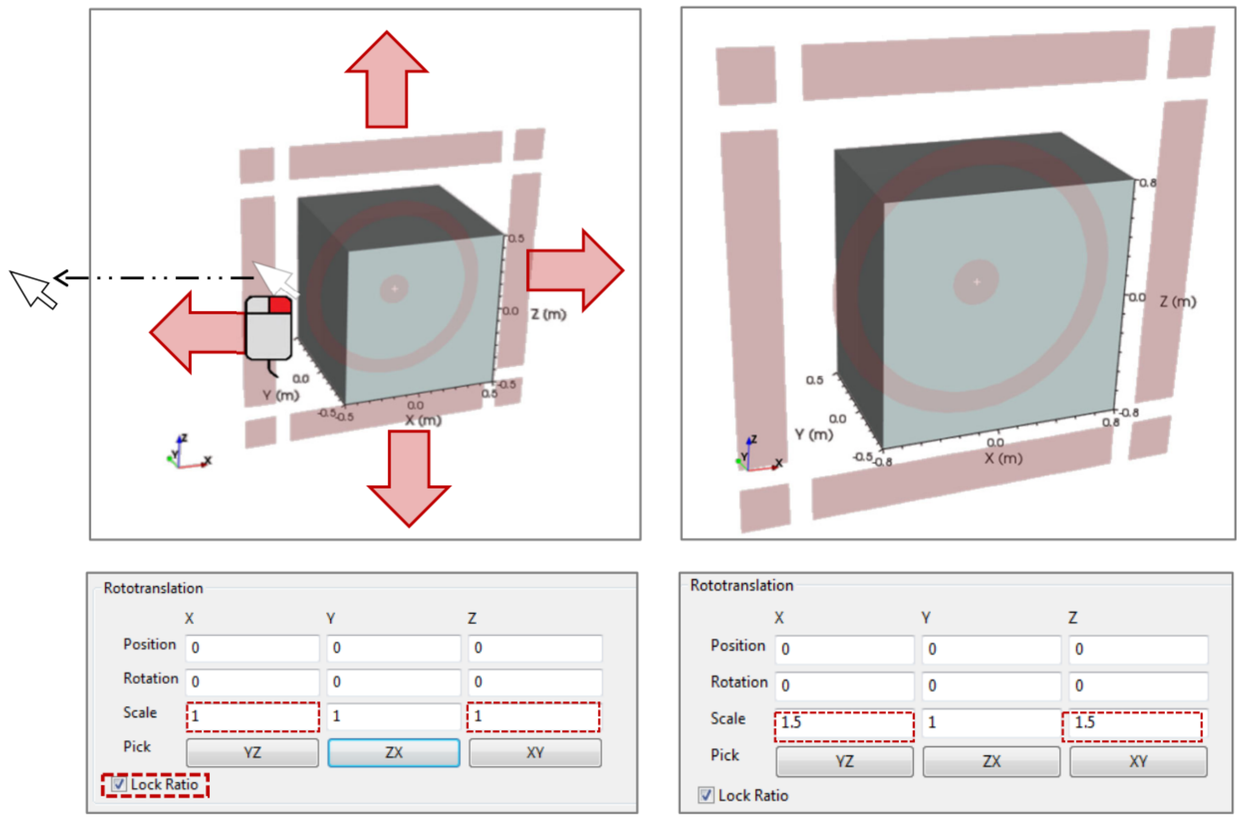


Figure 27.6: Example of manually stretching of a cube in two dimensions, with Lock Ration active

In the following paragraphs it will be shown how to insert various objects in the 3D scene. Each object has its position, its rotation, and its size, independently editable from other objects. Through the panel all the objects are treated as a single group. In the following example in the scene there is only one cube, but if more objects were inserted into the scene each of them will be subjected to changes made through the table. In the following example 5 spheres are placed in the scene (Figure 27.7).

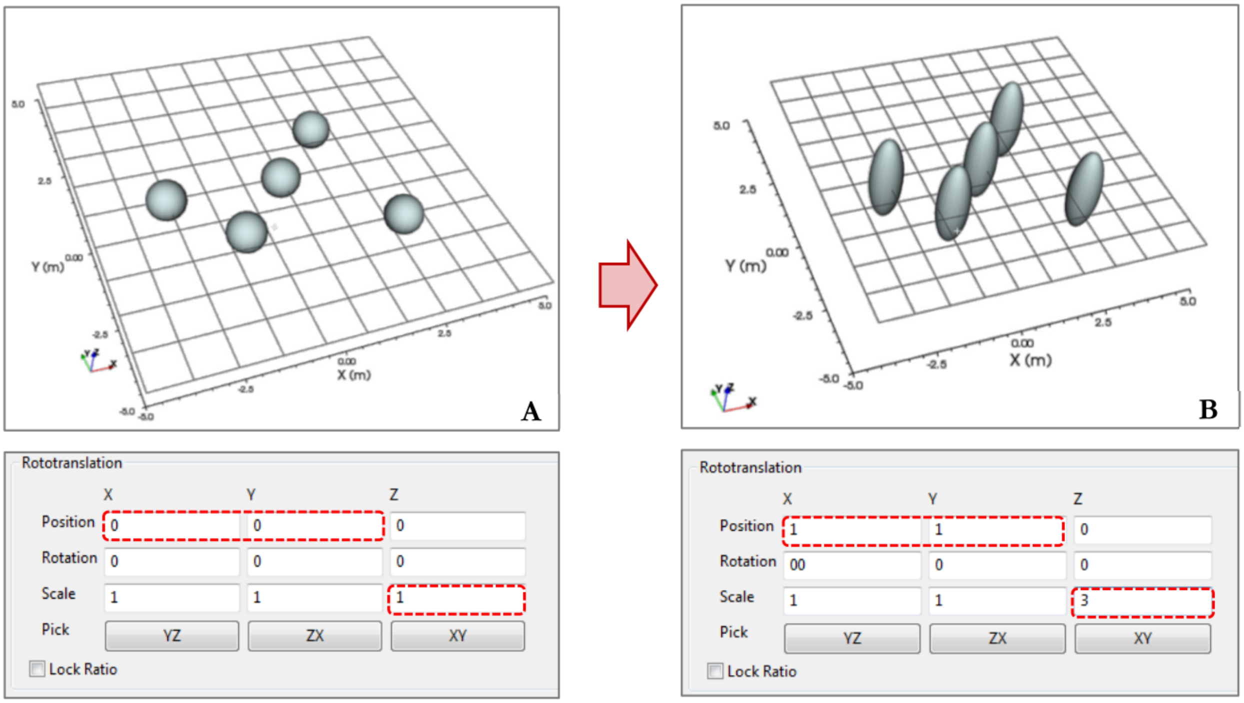


Figure 27.7: In A, objects in original position; in B the same objects after a change of position and scale in Scene panel

From image A to image B the position changes from 0 to 1 for the X and Y axes, and the Z axis is scaled by a factor of 3. The resulting scene is shifted and stretched along Z axis, thereby turning the spheres into 5 ovoid shape objects. But the relative position between the objects did not change.

CHAPTER

TWENTYEIGHT

COLOUR SCALE

Through this option the colour scale used to visualize the measurement points and the *Resistivity/Conductivity/IP/Sensitivity Models* can be customized (Figure 28.1).

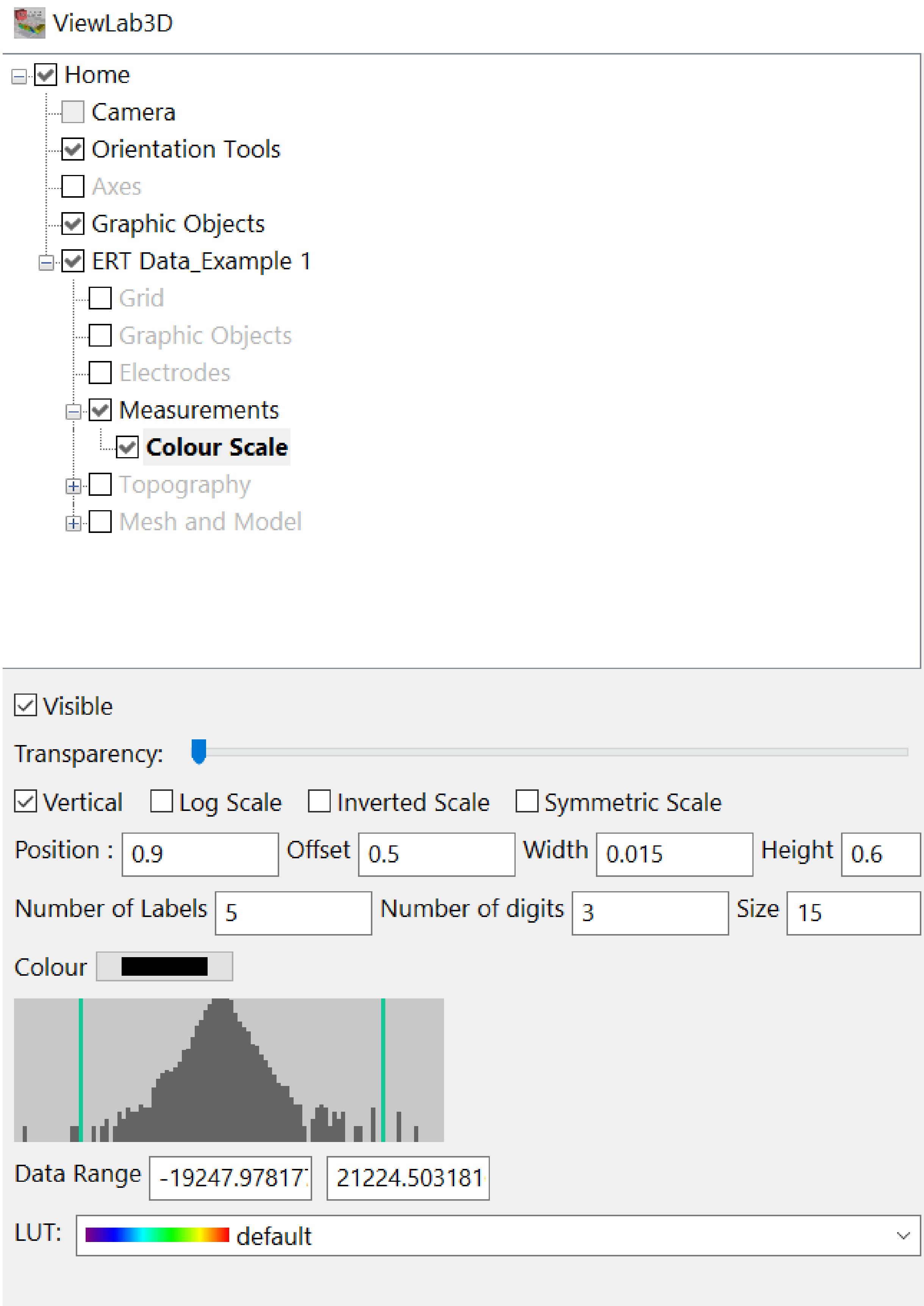


Figure 28.1: Colour Scale panel

28.1 Visible

Controls the colour scale visibility in the 3D space (Figure 28.2).

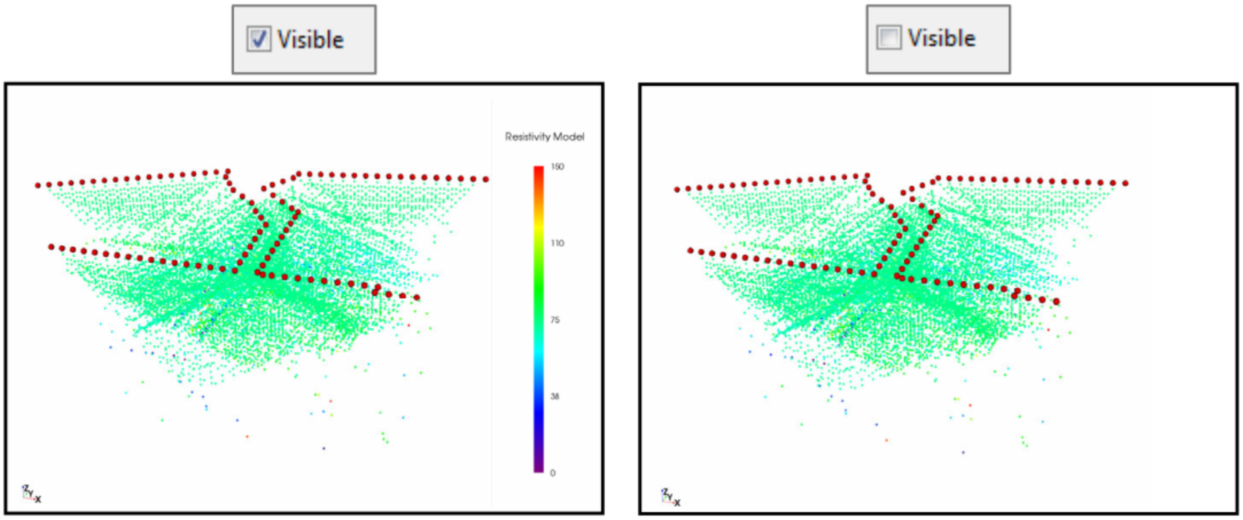


Figure 28.2: Setting of visibility of colour scale in measurement node

28.2 Transparency

This slider sets the transparency of the colour scale (Figure 28.3). As the slider shifts to the right, the colour scales become increasingly transparent.

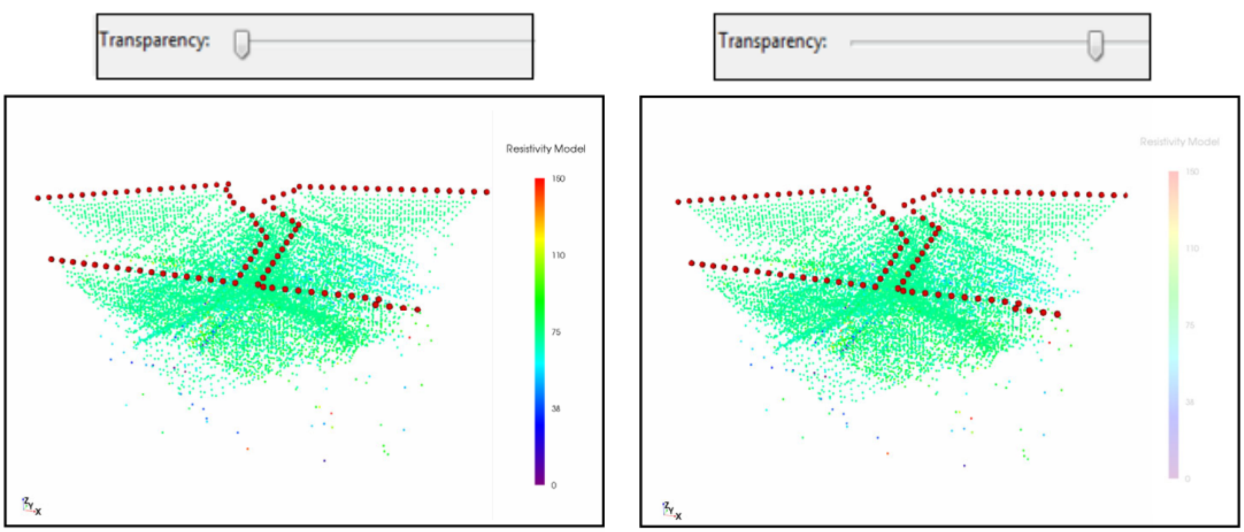


Figure 28.3: Different levels of transparency of the colour scale.

28.3 Vertical

Select the position of the colour scale. If checked, the colour scale is in vertical position; otherwise it is displayed horizontally in the 3D space. (Figure 28.4)

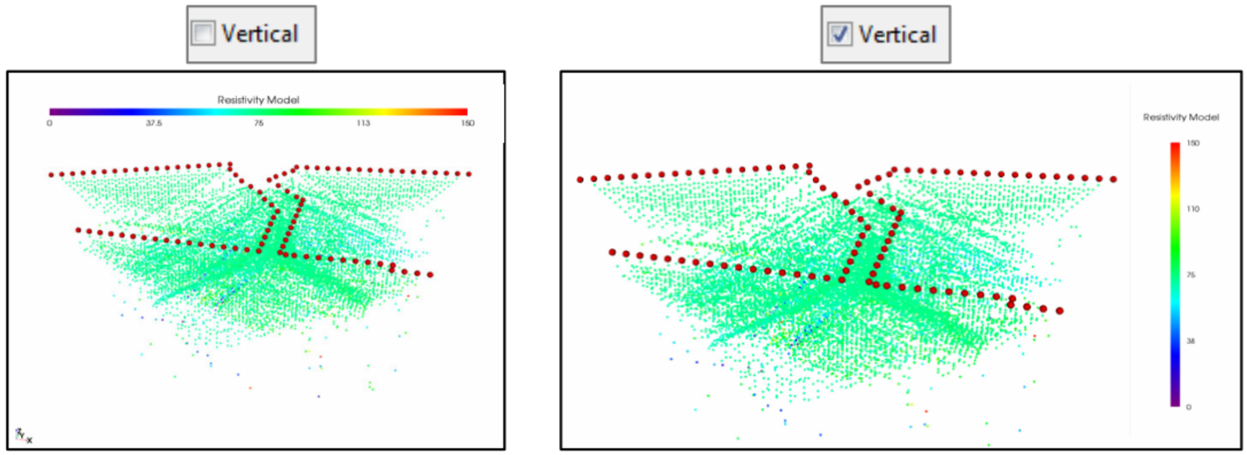


Figure 28.4: Colour Scale in horizontal and vertical position

To better appreciate the other options in the following examples the colour scale refers to the *Resistivity Model* and not to measurement points.

28.4 Log Scale

Turns the scale from linear to logarithmic (Figure 28.5). The colour of the scale are the same, but the step values are different (the number at the centre of the bar is 36 in logarithmic scale and 75 in linear scale). Consequently, also the representation mode of resistivity model changes and in some cases it helps to better appreciate the resistivity variation of interest.

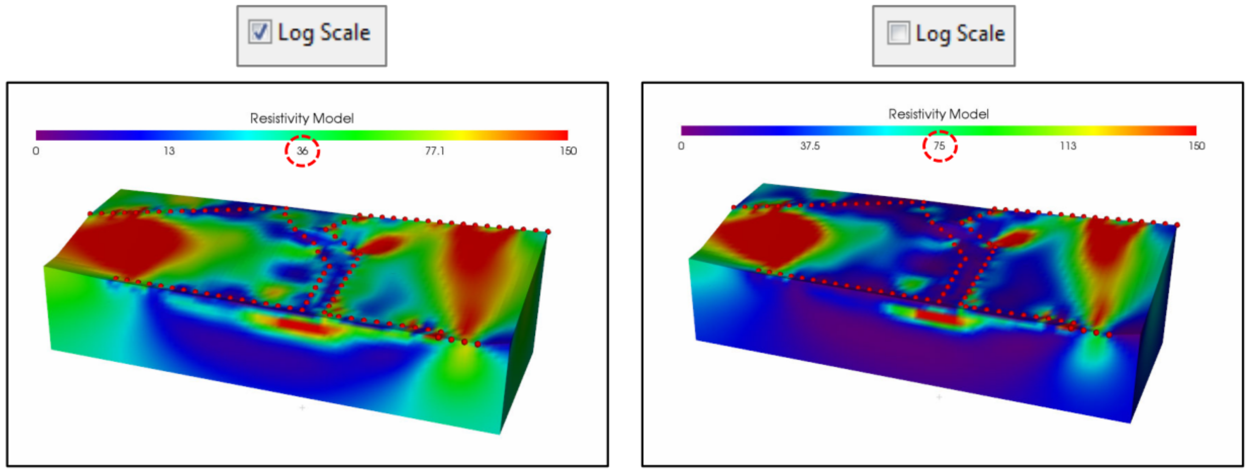


Figure 28.5: Example of logarithmic scale, on left, and linear scale, on the right

28.5 Inverted Scale

Inverts the colours inside the colour scale, so the colour associate with the maximum value will be associate to minimum value and vice versa. Consequently, the representation of the resistivity model also changes. In the following example, in not-inverted colour scale the minimum value corresponds to the purple colour; in the *Inverted Scale* to red (Figure 28.6).

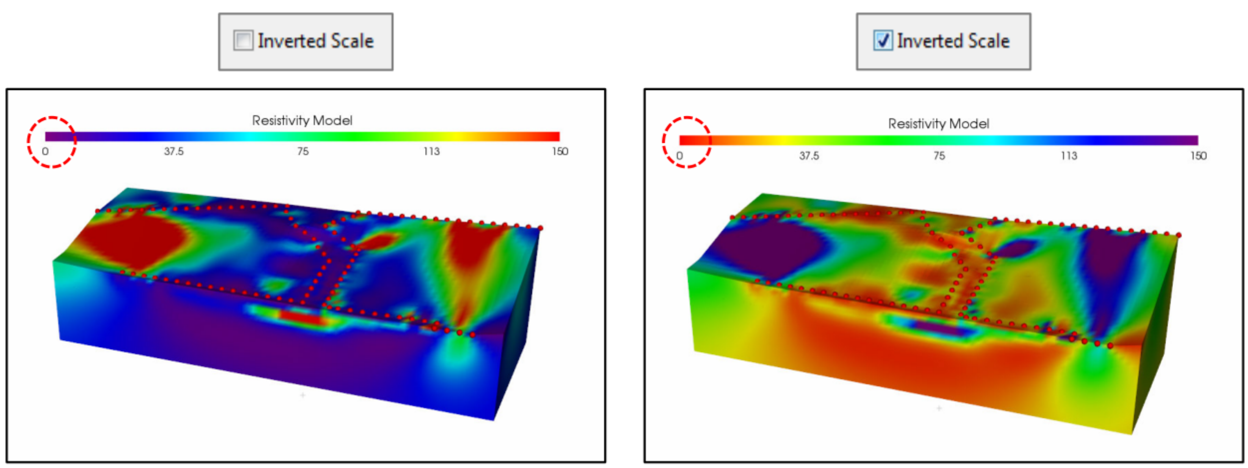


Figure 28.6: Example of inverted colour scale

28.6 Symmetric scale

Checking this box makes the selected colour scale become symmetrical with respect to the centre (Figure 28.7).

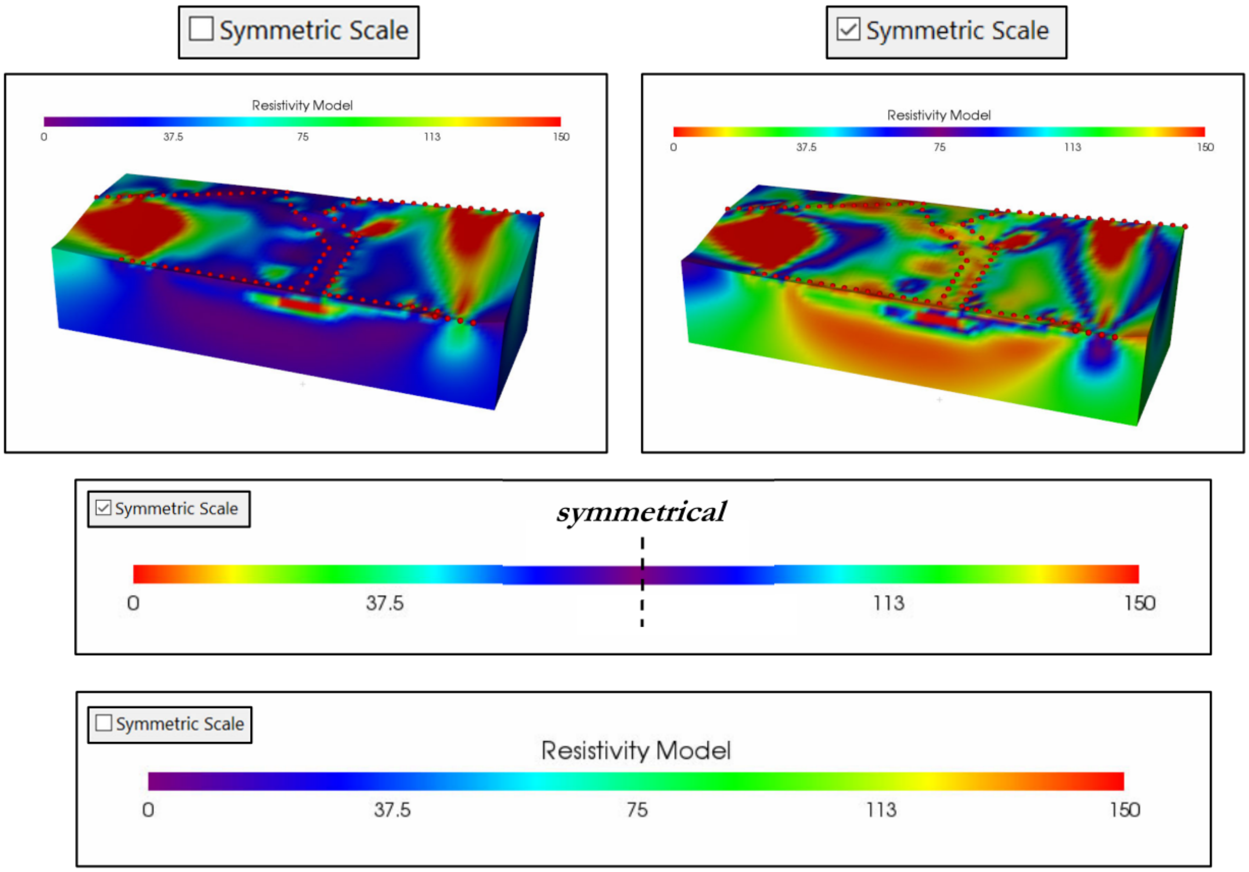


Figure 28.7: Example of Symmetrical colour scale

28.7 Position

Of the colour scale in the scene can be chosen. Typing the desired number (from 0 to 1) in the proper box or using the slider, which appears when positioning the mouse on the box (without clicking) will move the colour scale laterally (vertical checkbox active) and from the top to the bottom if the horizontal mode is active (Figure 28.8).

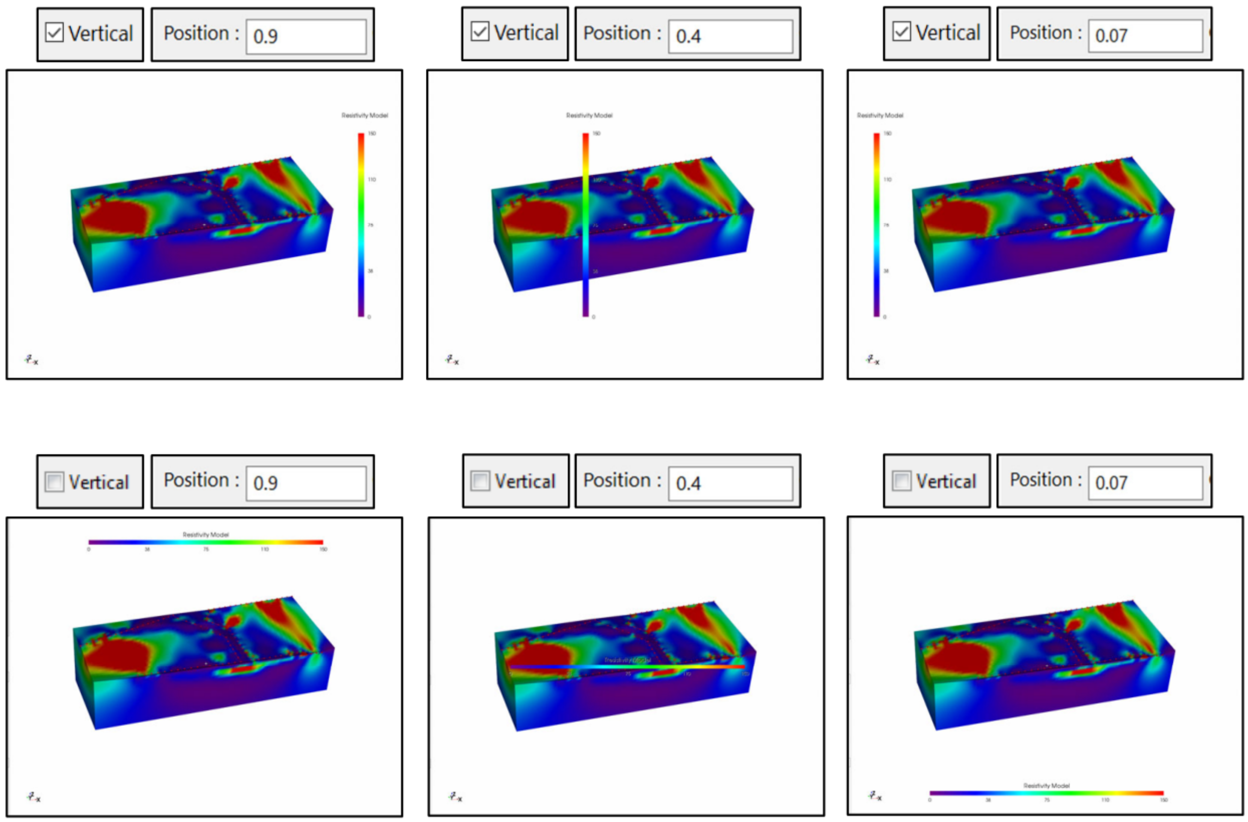


Figure 28.8: Example of different positions of vertical and horizontal colour scale in the 3D scene

28.8 Offset

Through this option the position of the colour scale in the direction perpendicular to those set through *Position* tool are set (Figure 28.9).

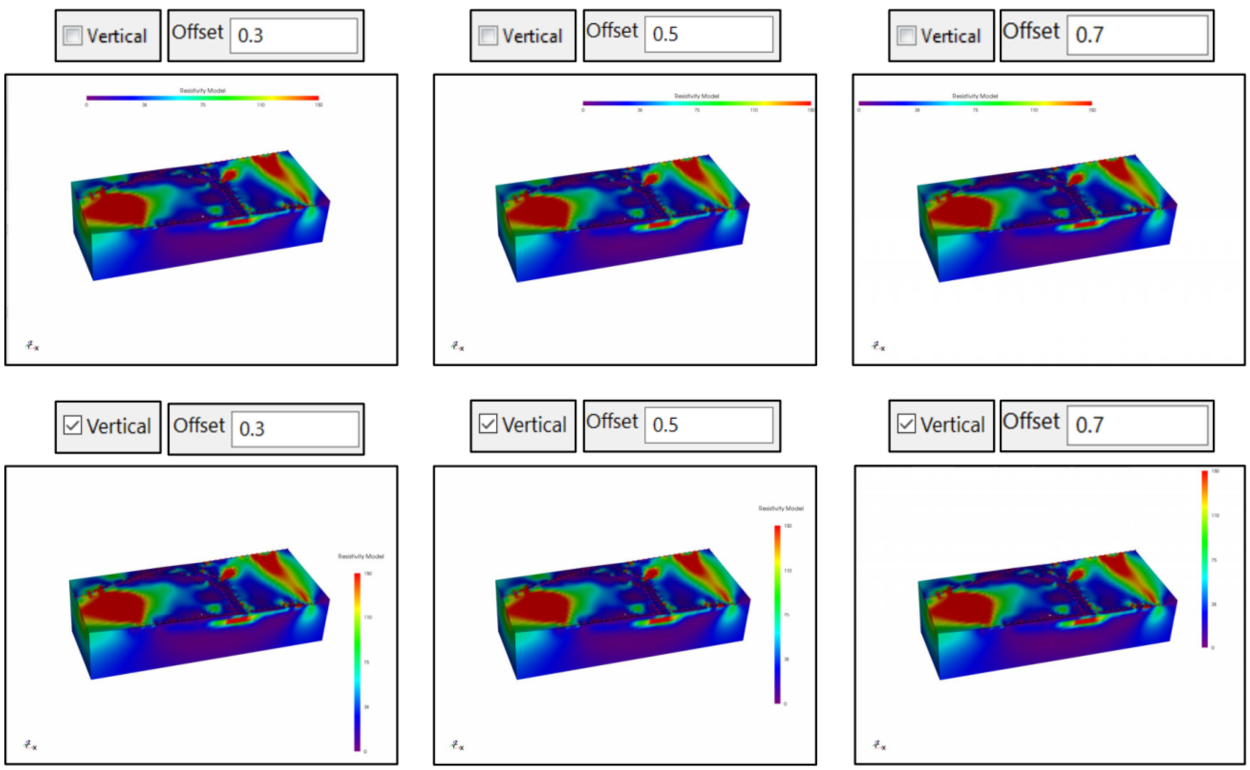


Figure 28.9: Example of different offset of vertical and horizontal colour scale in the 3D scene

28.9 Width

Choose the width of the colour scale. Typing the desired number (from 0 to 1) in the proper box or using the slider, which appears when positioning the mouse on the box (without clicking). Two example of a different width are shown in Figure 28.10.

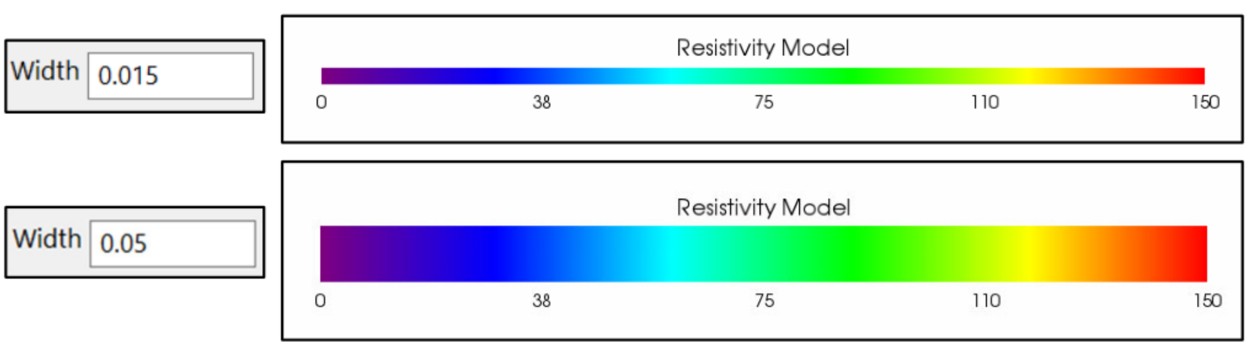


Figure 28.10: Example of different colour scale widths

28.10 Height

Choose the height of the colour scale. Typing the desired number (from 0 to 1) in the proper box or using the slider, which appears when positioning the mouse on the box (without clicking). Two example of a different width are shown in Figure 28.11.

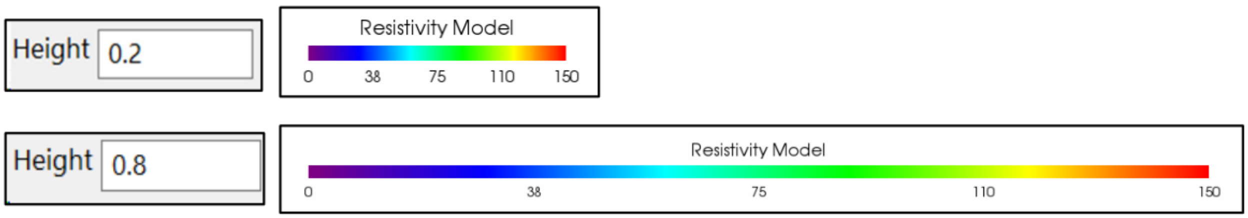


Figure 28.11: Example of different colour scale heights

28.11 Number of labels

Chooses how many labels to display on the colour scale, as it is shown in the following example (Figure 28.12).

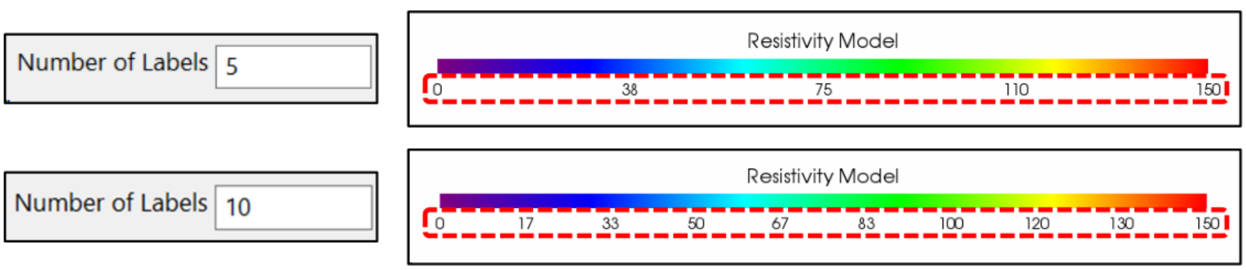


Figure 28.12: Example of different colour scale number of labels

28.12 Number of digits

Choose how many digits to display in the labels of the colour scale, as it is shown in the following example (Figure 28.13).

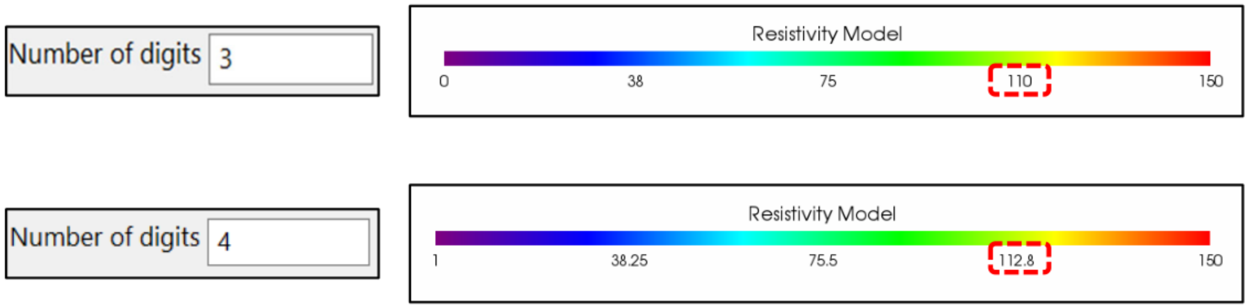


Figure 28.13: Example of different number of digits of the labels of the colour scale

28.13 Size

Scales the size of labels and the title of the colour scale, as it is shown in the following example (Figure 28.14).

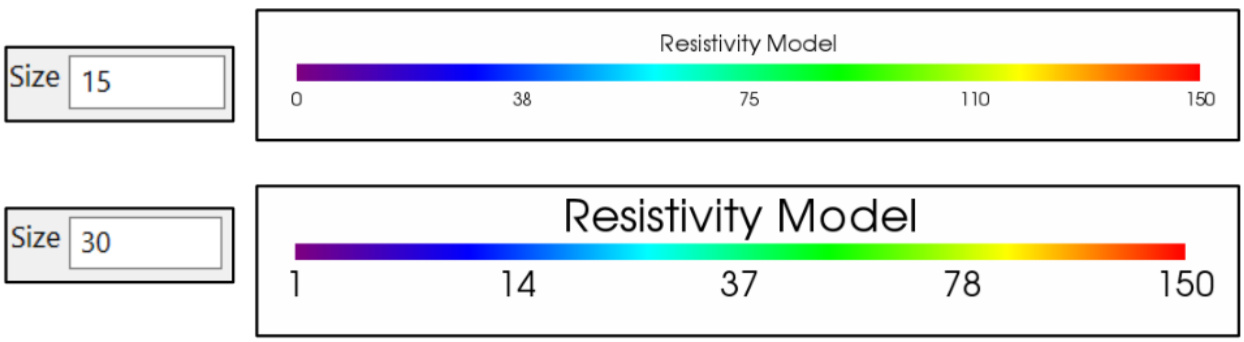


Figure 28.14: Example of different size of labels and title of the colour scale.

28.14 Colour

Selects the colour of labels and the title. It is black by default and it turns in red in the following example (Figure 28.15).

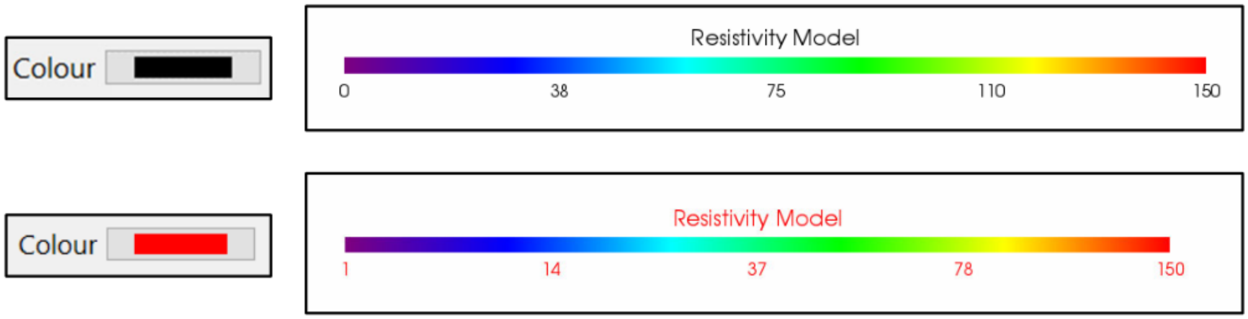


Figure 28.15: Example of different colour of labels and title of the colour scale.

28.15 Data range

Allows to set the minimum and the maximum values of the colour scale. Typing the desired values in the proper boxes, or using the slider which appears when positioning the mouse on the box (without clicking).

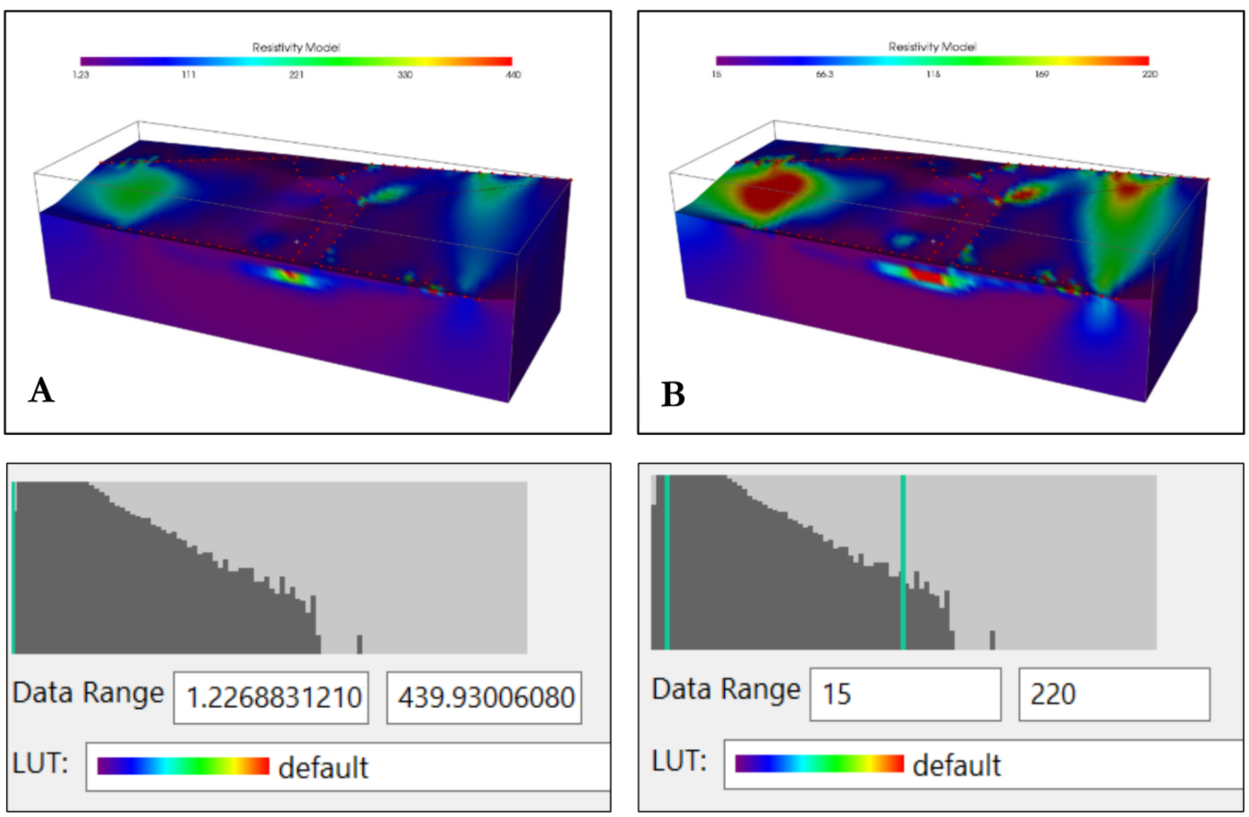


Figure 28.16: Different data range displayed; in A the data range of Colour scale is 1.22-440 (the entire data set), too wide to appreciate the distribution of the physical property in 3D model; in B the limits of colour scale are 15-220 and the Resistivity distribution is more visible

Automatic minimum and maximum values can be used keeping the "Auto" flag active.

To help the user to set the most appropriate values, a histogram shows the amount of data with the colours distribution. When changing the extreme numbers of the colour scale a low-cut and high-cut threshold bars will appear in the histogram in correspondence of the new minimum and maximum set values. This allows to better visualize the distribution of the physical property of the model (Figure 28.17).

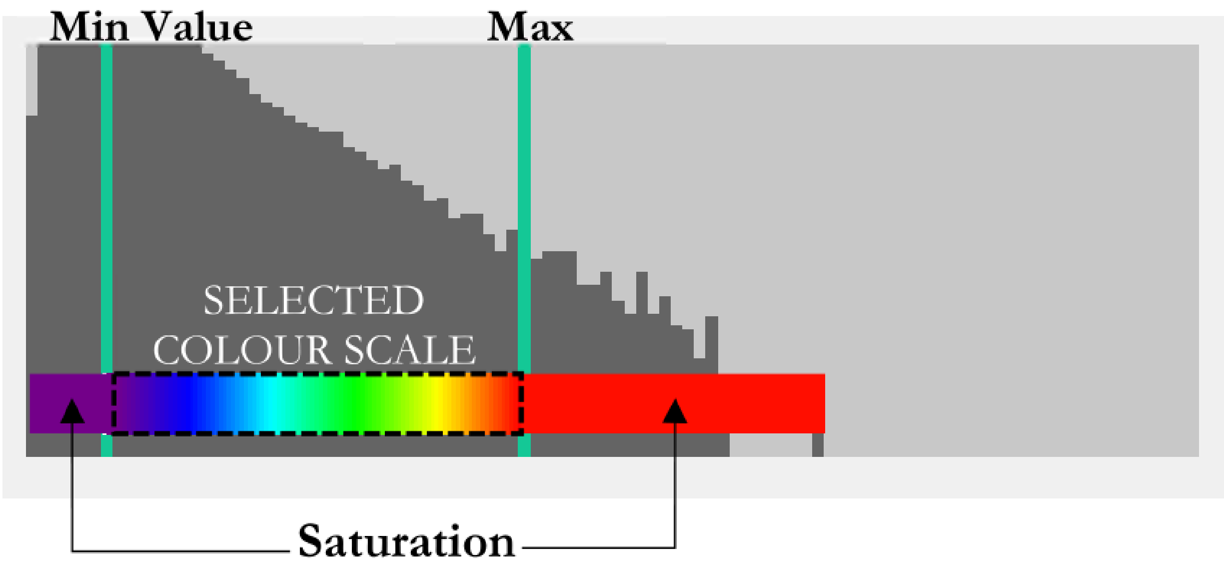


Figure 28.17: Saturation of Colour Scale in function of minimum and maximum values setting

The selected colour scale occupies the portion of histogram between the two bars, where most of the data are. Outside these cut-off bars the colour bar goes into saturation; this mean that all data from the *Max Value* until the right end of the histogram are of the same colour (red in the example), without any distinction. The same goes for data from the *Min Value* until the left end of the histogram, which are all purple in this case (Figure 28.17).

28.16 LUT

Allows to choose the colours of the scale. Figure 28.18 shows the possibilities available by default. These colour scales are generated from files loaded from the "Colourmap" directory, which is in the installation folder. Each *.lut file correspond to one colour bar in this list. This allows the user to delete or create new ones. You will have to re-start *ERTLab Studio* to access the new scale.

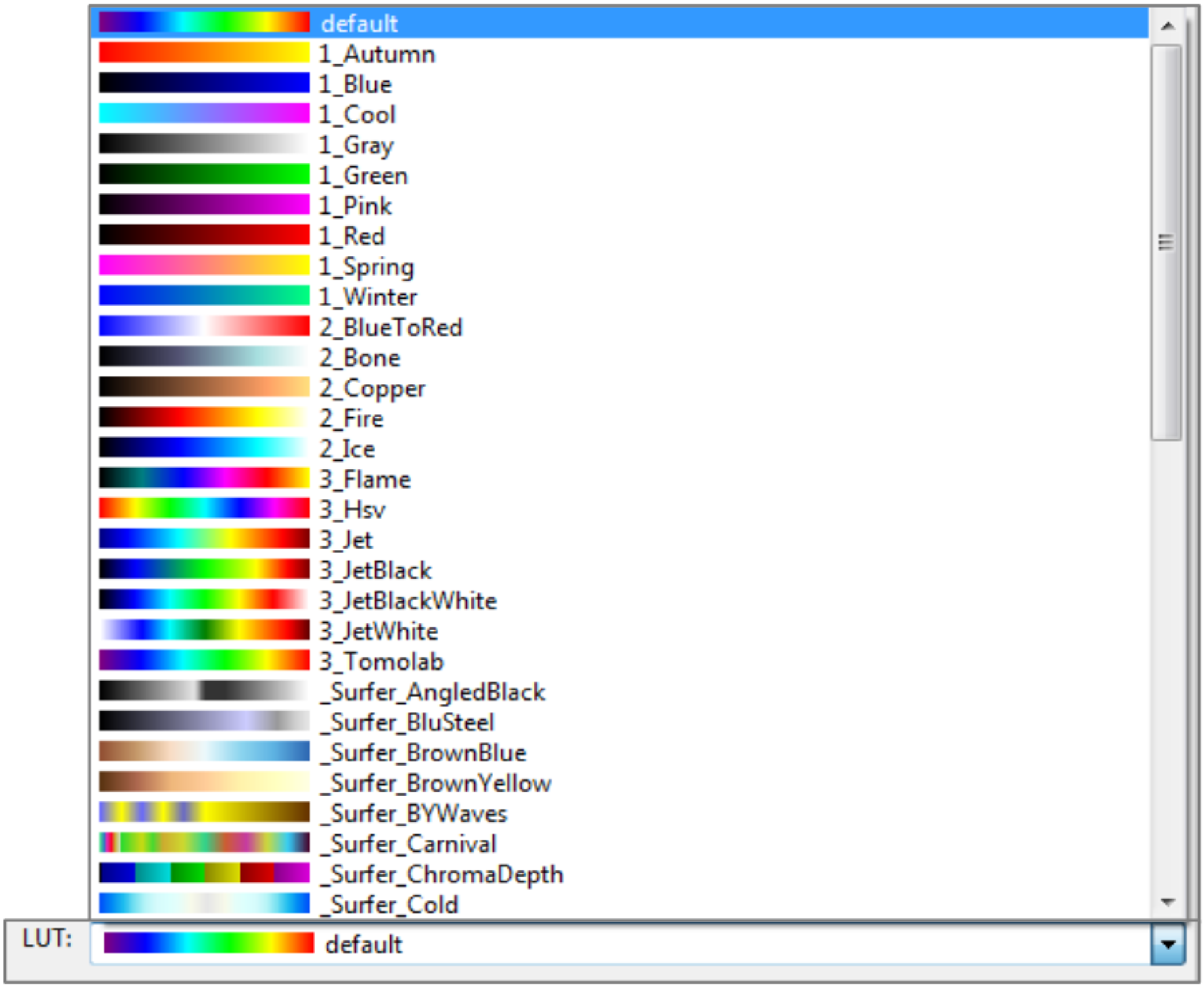


Figure 28.18: Example of some colours available for the colour map

The colour scale can be edited pressing the “Edit” button

Figure 28.19: Colour scale editing

The colour scale is usually build starting from a limited number of nodes (see black markers in the figure), interpolating the colours between them. In the editing panel it is possible to change the color related each node, and to change the node position. It is also possible to "Insert" a new node, or to "Delete" the current node. Pressing the "Import" or "Export" buttons on the bottom of the panel it is possible to save/load the color scale to/from a text file. Finally it is possible to save the modified color scale with the buttons "Save" (to overwrite) or "Save As" (to create a new scale, that will be available restarting the software).

CHAPTER TWENTYNINE

CENTRE CAMERA TO THIS

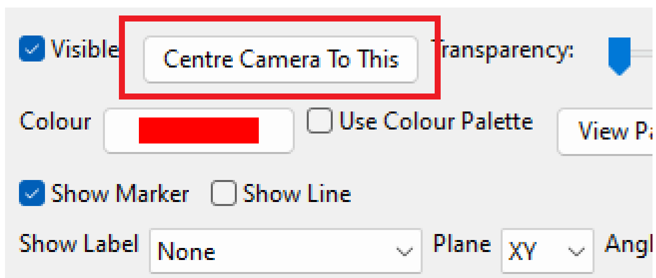


Figure 29.1: Tool "Centre camera" in the electrodes node

When more than one item is loaded (graphic objects, images, electrodes, topography . . .) it is possible that the camera remains focused on one of these items and the others are outside the scene. With this tool it is possible to centre the camera to the desired object. In the following example a cube is in the scene. With the *Auto Centre* button (see Auto Centre section) the camera zooms automatically to include all the object in the scene. If one or more objects are far from the others it is possible that one or more items appears too small to be visualized. In fact, in the example the cube is too far from the acquisition area to make the electrodes visible (Figure 29.2). With this tool it is possible to focus on what is in that moment more important to see.

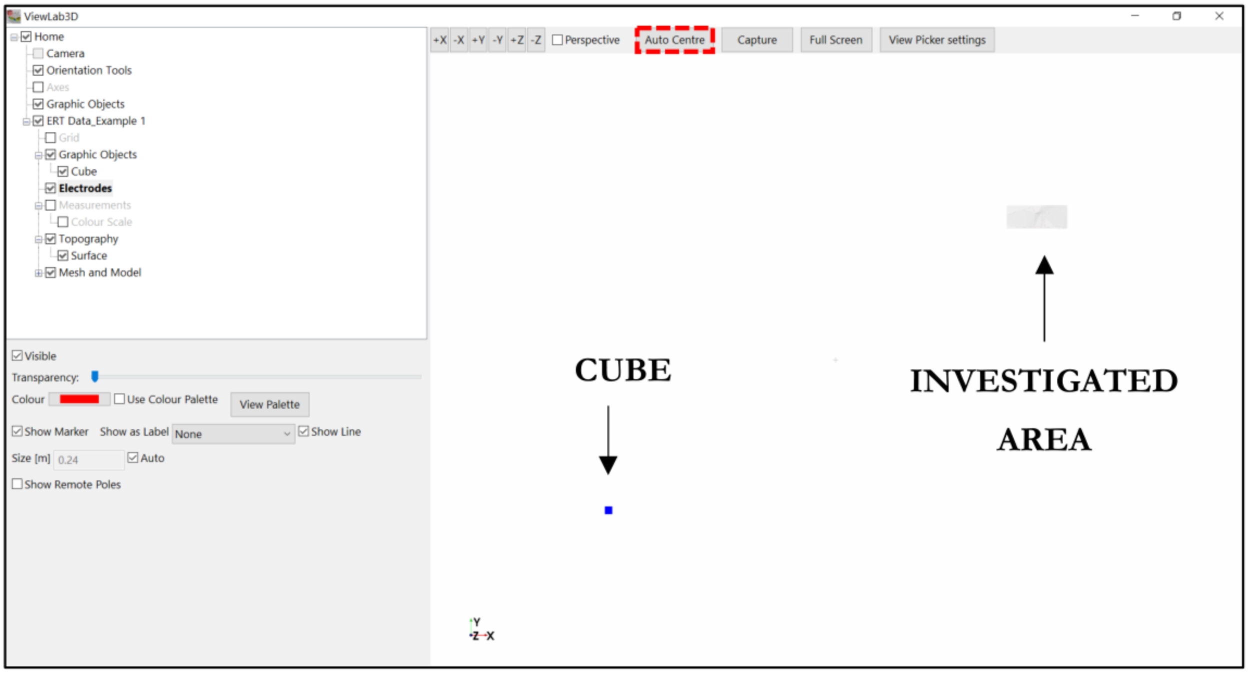


Figure 29.2: With "Auto Centre" button the cube and the investigated area are included in the scene, but both of them are too small to be appreciate

Through *Centre Camera to This* it is possible to centre the camera to the electrodes (Figure 29.3).

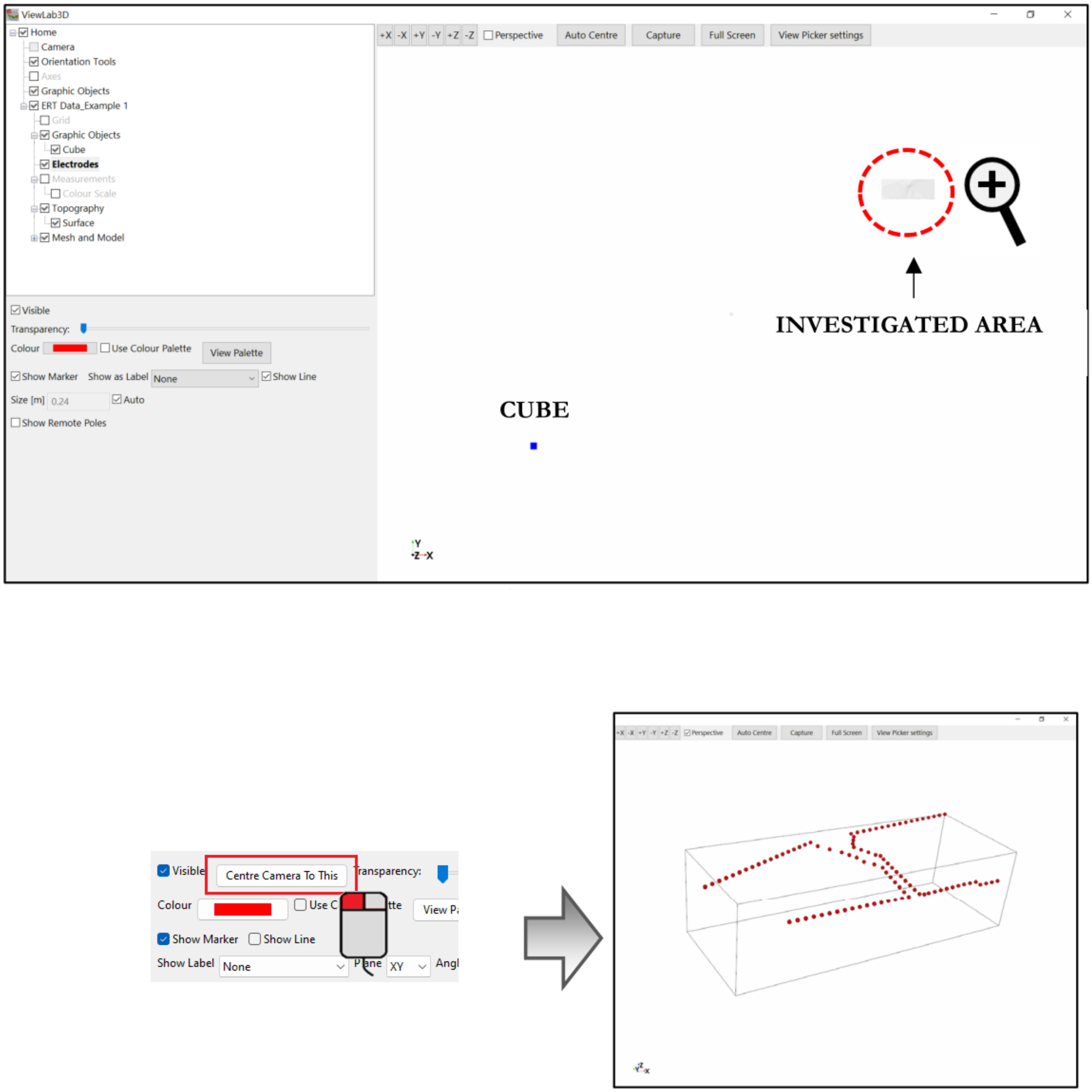


Figure 29.3: Centre camera to the Electrodes

In the same way, it is possible to centre the camera on the cube, clicking with the right mouse button in the proper node (Figure 29.4).

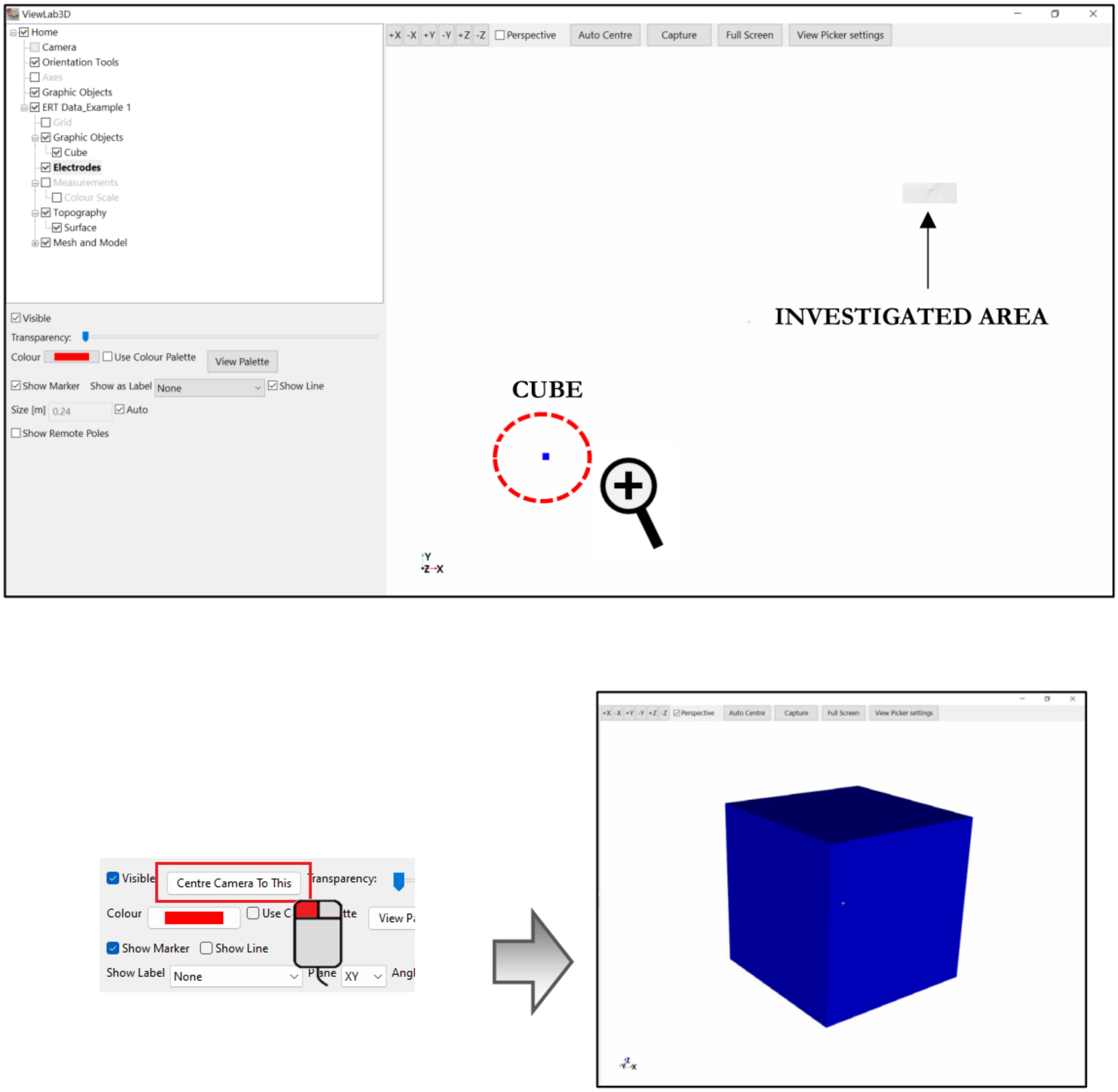


Figure 29.4: Centre Camera to the cube

CHAPTER

THIRTY

MATHEMATICAL NOTATIONS

30.1 Linear

- **Individual Value**: the returned values of this notification consist in their simple numerical sequence inserted.

0 2 5 6 10	→	0 2 5 6 10
(Value Value Value Value Value)		

Figure 30.1: Individual Value

- **Stepped Values**: specify the initial and final values of the desired sequence and the step between two successive values inbetween separated by colons.

1:1:5	→	1 2 3 4 5
(Initial value: Step: End value)		

Figure 30.2: Stepped Values

- **Equivalent Notation**: it is not necessary to insert the ":" between the step and the maximum value.

1:1 5	→	1 2 3 4 5
(Initial value: Step End value)		

Figure 30.3: Equivalent Notation

- **Stepped and Single Values**: a combination of the two notations described above. It provides a sequence of numbers between a given maximum and a minimum value, with a provided step, and it adds other given numbers at the end of the sequence.

0:2 10 11 18 → 0 2 4 6 8 10 11 18

(Initial Value: Step End_Value Value Value)

Figure 30.4: Stepped and Single Values

- **Combined Stepped Values List**: it gives a sequence of numbers between a given maximum and a minimum value with two or more different steps between them.

1:1 5:3 20 → 1 2 3 4 5 8 11 14 17 20

(Initial value: First step: Intermediate value: Second step: End value)

Figure 30.5: Combined Stepped Values List

- **Numeric**: with this notation is possible to choose the minimum and maximum values of the series and how many elements of the series visualize between the two extreme values, using the # symbol.

2 #4 10 → 2 4 6 8 10

(Initial value #number of intermediate value to insert End_Value)

Figure 30.6: Numeric

30.2 Logarithmic

With this notation it is possible to enter the minimum and maximum value of the series to represent and a series of intermediate values with increasing logarithmic trend. To achieve this use a step value multiplied by a "growing factor" (to apply from the third value onwards).

1:1*1.5 10 → 1 2 3.5 4.31 5.75 9.125 10

(Initial Value: Step* Growing Factor End-value)

Figure 30.7: Logarithmic

For example, in this case the step is 1 and the incremental factor is 1.5., resulting in:

First value		1
Second Value	First value (1) + Step (1)= 1+1	2
Third Value	Second Value (2) + Step(1) * Growing Factor (1.5) = 2+1*1.5	3.5
Fourth Value	Third Value (3.5) + "Grown Step" (1.5) * Growing Factor (1.5) = 3.5 + 1.5*1.5	5.75
	… and so on.	

30.3 Minimum-Maximum values

This notation uses automatically the maximum and the minimum value of a dataset and gives the possibility to choose how many intermediate values to insert between the two extreme values, as shown in the following example:

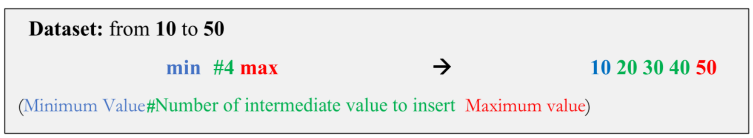


Figure 30.8: Minimum-Maximum values

30.4 Summary chart

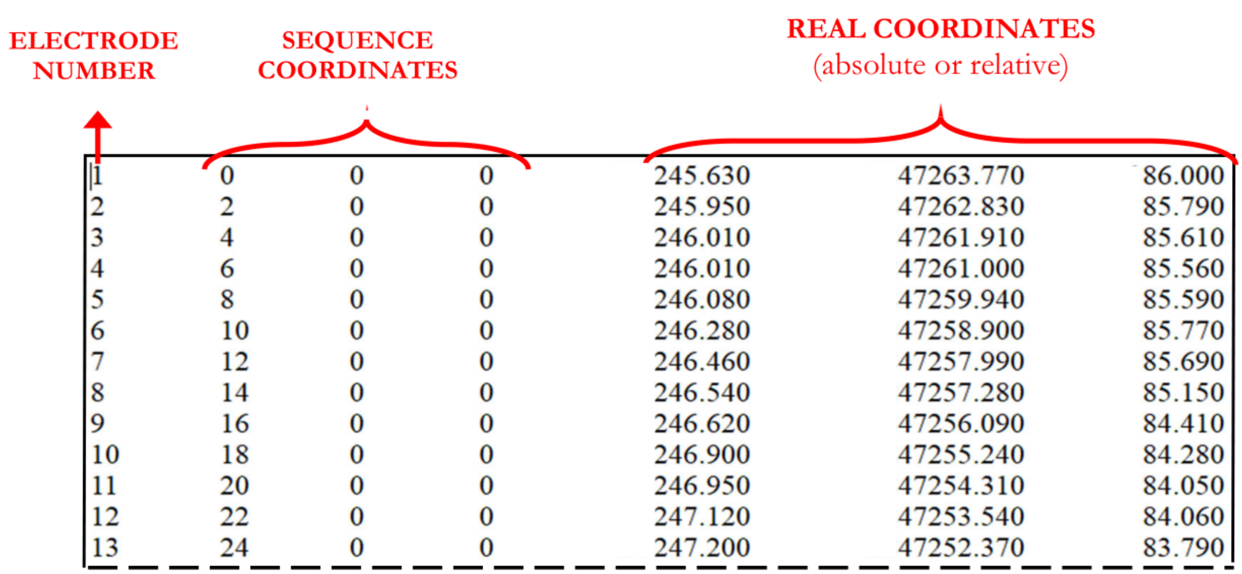
Notation type	**Notation**	**Value**
Explicit, Single Values	0 2 5 6 10	0 2 5 6 10
Linear, Stepped Values	1:1:5 1:1 5	1 2 3 4 5
Linear, Stepped Values	1:1 5	1 2 3 4 5
Linear, Stepped Values + Single Values	0:2 10 11 18	0 2 4 6 8 10 11 18
Linear, Combined stepped Value	1:1 5:3 20	1 2 3 4 5 8 11 14 17 20
Numeric	1#4 10	1 2 4 6 8 10
Logarithmic	1:1*1.5 10	1 2 3.5 4.31 5.75 9.12

CHAPTER

THIRTYONE

CONVERSION TABLE

Each Syscal *.BIN file (*Load Bin* button) can be associated to a **conversion table** to assign coordinates (absolute or relative) to the electrodes to which the measurements are associated to. Without the conversion table, data will be loaded with the coordinates used for the sequence acquisition. The conversion table is a 7-column .txt file (Figure 31.1):



1	0	0	0	245.630	47263.770	86.000
2	2	0	0	245.950	47262.830	85.790
3	4	0	0	246.010	47261.910	85.610
4	6	0	0	246.010	47261.000	85.560
5	8	0	0	246.080	47259.940	85.590
6	10	0	0	246.280	47258.900	85.770
7	12	0	0	246.460	47257.990	85.690
8	14	0	0	246.540	47257.280	85.150
9	16	0	0	246.620	47256.090	84.410
10	18	0	0	246.900	47255.240	84.280
11	20	0	0	246.950	47254.310	84.050
12	22	0	0	247.120	47253.540	84.060
13	24	0	0	247.200	47252.370	83.790

Figure 31.1: Example of conversion table

The conversion table can also be a 8-column file, in that case the first two columns are used to give to the electrodes a "group" and "id" identification.

```
Cable1   1   0   0   0   852.48   152.27   63.33
Cable1   2   1   0   0   857.94   152.20   63.03
Cable1   3   2   0   0   863.40   152.13   62.90
Cable1   4   3   0   0   868.86   152.07   63.18
Cable1   5   4   0   0   874.32   152.00   63.42
Cable1   6   5   0   0   879.78   151.93   63.06
```

(continues on next page)

(continued from previous page)

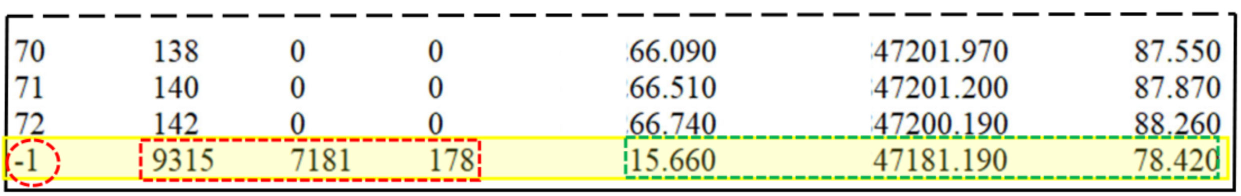
```
Cable2   1   6   0   0   885.24   151.87   63.27
Cable2   2   7   0   0   890.70   151.80   62.89
Cable2   3   8   0   0   896.16   151.73   62.53
Cable2   4   9   0   0   901.62   151.66   62.52
Cable2   5  10   0   0   907.08   151.60   63.36
Cable2   6  11   0   0   912.54   151.53   63.03
```

If the electrode coordinates are not matching correctly the ones provided with the conversion table then it is ignored. So it can be useful to export the electrode table after loading the raw BIN file (see section Export) as a start point to create the conversion table.

ERTLab Studio automatically reads the conversion table when a *.BIN file is loaded, provided that the two files have the *same name*:

In case of acquisition with a **remote pole** it is possible to operate in two different ways:

- After loading the project: check  in the REM column of the corresponding electrode (in electrode table, see section Table).
- Before loading the project: insert the remote pole in the conversion table, associating it at the ELECTRODE NUMBER = **-1**; so that ERTLab *Studio* can automatically identify it as a remote pole (Figure 31.2).



70	138	0	0	66.090	47201.970	87.550
71	140	0	0	66.510	47201.200	87.870
72	142	0	0	66.740	47200.190	88.260
-1	9315	7181	178	15.660	47181.190	78.420

Figure 31.2: Insertion of remote pole in the conversion table

Only the real coordinates are used (in green in Figure 31.2) for remote poles, so the sequence coordinates (in the red box in Figure 31.2) of a remote pole can be any number (like 9315, 7181, 178 as in the example, or 0,0,0) because anyway they will be ignored.

CHAPTER

THIRTYTWO

DATA FILE

All the information related an **ERTlab Studio** project are saved in a "*.data*" file. It is a plain text file, so it can be also opened with "*Notepad*" to check its content.

ERTLab employs keywords (tags) to indicate configuration information. Each keyword starts with the **#** symbol, and end with the **=** symbol.

The comment character is **!**. Any information placed between ! and the end of a file line is ignored. Blank lines are also ignored.

If it is provided an unknown keyword, than the full line is skipped.

32.1 Electrodes

The Electrode format requires an **#elec_start** keyword to indicate the start of the electrode listing and an **#elec_end** keyword to indicate the end of the list.

Electrodes input format can be included in the data file by using the appropriate configuration tags. The configuration information may include:

Flags:

- **#elec_no_cable=** indicates the presence of cable identifiers (1 present; -1 not present)

Column identifiers:

- **#elec_cable_col=** cable group (cable number)
- **#elec_id_col=** electrode number
- **#elec_x_col=** electrode X location
- **#elec_y_col=** electrode Y location
- **#elec_z_col=** electrode Z location
- **#elec_elev_col=** electrode Z terrain elevation

- **#elec_type_col=** type of electrode (1 borehole; 2 surface; -1 remote borehole; -2 remote surface)
- **#elec_tx_col=** flag to identify transmitting electrode (1 yes; 2 no)
- **#elec_rx_col=** flag to identify receiving electrode (1 yes; 2 no)
- **#elec_skip_col=** flag to identify skipped electrode (1 yes; 2 no)
- **#elec_roll_col=** flag to identify roll electrode (1 yes; 2 no)

The value **-1** stands for column not provided/information not present. If the electrode type is not specified in the input file, ERTLab will derive it for each electrode. "Remote" flags will be attributed to the cables having one or two electrodes. If the electrode z terrain elevation is not specified in the input file, ERTLab will derive it for each electrode.

See the small example below:

```
#elec_no_cable= 1
#elec_cable_col= 1
#elec_id_col= 2
#elec_x_col= 3
#elec_y_col= 4
#elec_z_col= 5
#elec_elev_col= 6
#elec_type_col= 7

#elec_start
! Cab ID        X       Y       Z     Elev    Type
1      1      1000     100      0      0      -2    ! Remote surface electrode
2      1     -1000     100     -5      0      -1    ! Remote borehole electrode
3      1      -100     100      0      0       2    ! Five electrode surface␣
↪cable
3      2       -50     100      0      0       2
3      3         0     100      0      0       2
3      4        50     100      0      0       2
3      5       100     100      0      0       2
4      1         0     100    -25      0       1    ! Five electrode well
4      2         0     100    -20      0       1
4      3         0     100    -15      0       1
4      4         0     100    -10      0       1
4      5         0     100     -5      0       1
#elec_end
```

32.2 Data

The Data format requires a **#data_start** keyword to indicate the start of the data listing and a **#data_end** keyword to indicate the end of the list.

Data input format can be included in the data file by using the appropriate configuration tags. The configuration information may include:

Flags:

- **#data_appres=** flag for data in terms of (1) resistance V/I or (2) apparent resistivity
- **#data_ip_scale=** scale factor for IP data

Column identifiers:

- **#data_id_col=** quadrupole number
- **#data_a_cable_col=** TX+ (A) electrode cable number
- **#data_a_elec_col=** TX+ (A) electrode id number
- **#data_b_cable_col=** TX- (B) electrode cable number
- **#data_b_elec_col=** TX- (B) electrode id number
- **#data_m_cable_col=** RX+ (M) electrode cable number
- **#data_m_elec_col=** RX+ (M) electrode id number
- **#data_n_cable_col=** RX- (N) electrode cable number
- **#data_n_elec_col=** RX- (N) electrode id number
- **#data_quad_x_col=** quadrupole estimated position (X)
- **#data_quad_y_col=** quadrupole estimated position (Y)
- **#data_quad_z_col=** quadrupole estimated position (Z)
- **#data_geom_fact_col=** quadrupole estimated geometric factor (K)
- **#data_i_curr_col=** field data TX/AB current (I)
- **#data_v_col=** field data RX/MN potential (V)
- **#data_res_col=** field data RX/MN resistance (V/I)
- **#data_ip_wind_col=** field data IP
- **#data_std_v_col=** field data potential standard deviation
- **#data_std_res_col=** field data resistance standard deviation
- **#data_std_ip_col=** field data IP standard deviation
- **#data_calc_v_col=** calculated potential (V)

- **#data_calc_res_col=** calculated resistance (V/I)
- **#data_calc_ip_col=** calculated IP
- **#data_calc_std_v_col=** calculated potential standard deviation
- **#data_calc_std_res_col=** calculated resistance standard deviation
- **#data_calc_std_ip_col=** calculated IP standard deviation
- **#data_skip_col=** flag to identify skipped data (0 valid measurement ; 1 filtered measurement)

The value **-1** stands for column not provided/information not present.

See the small example below:

```
#data_id_col= 1
#data_a_cable_col= 2
#data_a_elec_col= 3
#data_b_cable_col= 4
#data_b_elec_col= 5
#data_m_cable_col= 6
#data_m_elec_col= 7
#data_n_cable_col= 8
#data_n_elec_col= 9
#data_res_col= 10
#data_ip_wind_col= 11
#data_std_res_col= -1
#data_std_ip_col= -1
#data_calc_res_col= -1
#data_calc_ip_col= -1
#data_calc_std_res_col= -1
#data_calc_std_ip_col= -1
#data_appres= 1
#data_ip_scale= 1000

#data_start
! ID C_A    A    C_B    B    C_M    M    C_N    N      Iab(mA)   Vmn(V)
1      1    1     4     1     4     2     2     1      0.1115     12.2
2      1    1     4     2     4     3     2     1      0.1122      1.5
3      1    1     4     3     4     4     2     1      0.1760      7.6
4      1    1     4     4     4     5     2     1      0.2114      4.4
5      3    1     3     2     3     3     3     4     -0.1130     10.1
6      3    2     3     3     3     4     3     5     -0.1220     44.7
7      3    1     3     3     3     2     3     4      0.1175     16.1
8      3    2     3     4     3     3     3     5      0.1625      3.3
#data_end
```

32.3 Topography

The topography information can be included in the DATA file file by using the appropriate tags. The Topography format requires an **#topo_start** keyword to indicate the start of the topography listing and an **#topo_end** keyword to indicate the end of the list. The configuration information may include:

Column identifiers:

- **#topo_x_col=** point X location
- **#topo_x_col=** point Y location
- **#topo_x_col=** point Z location

See the small example below:

```
#topo_x_col=1
#topo_y_col=2
#topo_z_col=3

#topo_start
!  X          Y          Z
100001     1100020     351
100010     1100001     357
100101     1100045     348
100007     1100106     355
100025     1100092     352
100076     1100060     358
100051     1100044     349
100021     1100036     350
100019     1100082     353
100005     1100011     355
#topo_end
```

32.4 Mesh and Model

This section includes the parameters described also in Run Mesh Generation .

The following parameters are used when generating the mesh grid:

- **#mesh_type=** Type of mesh, only mesh type 0 is available
- **#dim_grid_x=** Interior Grid Element X Size (m)
- **#dim_grid_y=** Interior Grid Element Y Size (m)

- **#dim_grid_z=** Interior Grid Element Z Size (m)
- **#foreground_x_min=** Mesh Minimum X Distance (m)
- **#foreground_x_max=** Mesh Maximum X Distance (m)
- **#foreground_y_min=** Mesh Minimum Y Distance (m)
- **#foreground_y_max=** Mesh Maximum Y Distance (m)
- **#foreground_z_min=** Mesh Minimum Z Value (m)
- **#foreground_z_max=** Mesh Maximum Z Value (m)
- **#pads_x_min=** Grid Left (-X) Pads (interior X grid units)
- **#pads_x_max=** Grid Right (+X) Pads (interior X grid units)
- **#pads_y_min=** Grid Back (-Y) Pads (interior Y grid units)
- **#pads_y_max=** Grid Front (+Y) Pads (interior Y grid units)
- **#pads_z_min=** Grid Bottom (-Z) Pads (interior Z grid units)
- **#pads_z_max=** Grid Top (+Z) Pads (interior Z grid units)
- **#flat_grid=** flag to create a flat grid (1 yes; 2 no)
- **#use_elec_in_topography=** flag to add electrodes Z terrain to topography (1 yes; 2 no)
- **#update_z_surface_electrodes=** flag to update elevation for Surface electrodes (1 yes; 2 no)
- **#flat_z_bottom=** flag to force the grid bottom to be flat (1 yes; 2 no)
- **#depth_of_investigation=** value to calculate the grid bottom from the top when topographic information is used
- **#adapt_grid_to_electrodes=** flag to adapt grid to electrodes (1 yes; 2 no)
- **#resistivity_model_type=** Type of resistivity model, only model type 0 is available
- **#background_res=** Background Resistivity (ohm m)
- **#ip_model_type=** Type of IP model, only model type 0 is available.
- **#background_ip=** Background Ip Value (mV/V)

To store in the DATA file the Mesh grid is requires also:

- a **#node_start** keyword to indicate the start of the data listing and a **#node_end** keyword to indicate the end of the node list.
- a **#model_start** keyword to indicate the start of the data listing and a **#model_end** keyword to indicate the end of the model list.

The following parameters help to correctly reading the previous lists:

- **#mesh_node_x_size=** number of nodes in the X direction

- **#mesh_node_y_size=** number of nodes in the Y direction
- **#mesh_node_z_size=** number of nodes in the Y direction
- **#mesh_node_x_col=** column for node X coordinates
- **#mesh_node_y_col=** column for node Y coordinates
- **#mesh_node_z_col=** column for node Z coordinates
- **#mesh_node_type_col=** column for node type value (1 foreground; 2 background)
- **#model_res_col=** column for resistivity (Res) model
- **#model_ip_col=** column for chargeability (IP) model
- **#model_sensitivity_col=** column for sensitivity model

```
#mesh_type= 0-Standard
#dim_grid_x= 0.5
#dim_grid_y= 0.5
#dim_grid_z= 0.5
#foreground_x_min= 0
#foreground_x_max= 47
#foreground_y_min= -0.5
#foreground_y_max= 0.5
#foreground_z_min= -7.84
#foreground_z_max= 0
#pads_x_min= 1 2
#pads_x_max= 1 2
#pads_y_min= 1 2
#pads_y_max= 1 2
#pads_z_min= 1 2
#pads_z_max= 0
#flat_grid= 1
#use_elec_in_topography= 2
#update_z_surface_electrodes= 2
#flat_z_bottom= 1
#depth_of_investigation= 7.84
#adapt_grid_to_electrodes= 2

#resistivity_model_type= 0-Standard
#background_res= 4.246
#ip_model_type= 0-Standard
#background_ip= 30.9
#mesh_node_x_size=99
#mesh_node_y_size=7
#mesh_node_z_size=19
```

(continues on next page)

(continued from previous page)

```
#mesh_node_x_col= 1
#mesh_node_y_col= 2
#mesh_node_z_col= 3
#mesh_node_type_col= 4
#model_res_col= 1
#model_ip_col= 2
#model_sensitivity_col= -1

#mesh_node_start
!               X       Y       Z       Type
-1      -1.5    0       2
-0.5    -1.5    0       2
0       -1.5    0       2
...
47      1.5     -8.84   2
47.5    1.5     -8.84   2
48      1.5     -8.84   2
#mesh_node_end

#model_start
!       RES     IP
76.275 30.9
66.416 30.9
52.751 30.9
...
3.461  30.9
3.401  30.9
3.325  30.9
#model_end
```

32.5 Forward Model

This section includes the parameters described also in Run Forward Model , in Run Inversion and in Run Sensitivity .

- **#pc_num_core=** number of (virtual) cores used during computation
- **#boundary_conds=** array of flags to specify the boundary conditions used on the 6 (-X +X -Y +Y -Z +Z) directions (0 Dirichlet or zero voltage boundaries; 1 Mixed; 2 Neumann or no-flow boundaries)
- **#forw_ssor_omega=** Omega (SQR preconditioning) value

- **#forw_tolerance=** Forward Solver Tolerance value
- **#forw_max_iterations=** Maximum Number of Forward iterations
- **#skip_tx_electrodes_with_no_rx=** flag to skip (1) or not (2) modelling of electrodes not used as receiver
- **#model_ip=** flag to model IP data also (1 yes; 2 no)

```
#pc_num_core= 7

#boundary_conds= 1 1 1 1 1 2
#forw_ssor_omega= 1.6
#forw_tolerance= 1e-007
#forw_max_iterations= 4000
#skip_tx_electrodes_with_no_rx= 2
#model_ip= 2
```

32.6 Inversion

This section includes the parameters described also in Run Inversion .

- **#rough_trials_iter=** array to set roughness trials at eache iteration
- **#constrain_to_starting=** Flag to Constrain to starting model (1 yes; 2 no)
- **#constrain_to_ref=** Flag to Constrain to reference model (1 yes; 2 no)
- **#max_num_iterations_rho=** Maximum Number Inversion Iterations (Rho)
- **#max_num_iterations_ip=** Maximum Number Inversion Iterations (IP)
- **#max_internal_pcg_rho=** Maximum Internal Inverse PCG Iterations (Rho)
- **#max_internal_pcg_ip=** Maximum Internal Inverse PCG Iterations (IP)
- **#inv_pcg_tolerance_rho=** Tolerance for Inverse PCG iterations (Rho)
- **#inv_pcg_tolerance_ip=** Tolerance for Inverse PCG iterations (IP)
- **#optimal_rough_factor_rho=** Tolerance on target minimum error (Res)
- **#optimal_rough_factor_ip=** Tolerance on target minimum error (IP)
- **#rough_multiplier_rho=** Multiplier for Changing Roughness Factor (Rho)
- **#rough_multiplier_ip=** Multiplier for Changing Roughness Factor (IP)
- **#initial_rough_rho=** Initial Roughness Factor (Rho)
- **#initial_rough_ip=** Initial Roughness Factor (IP)

- **#x_rough_weight=** Value for Roughness Param X Weights
- **#y_rough_weight=** Value for Roughness Param Y Weights
- **#z_rough_weight=** Value for Roughness Param Z Weights
- **#res_noise_percent=** Data Percent Errors [%] (Res)
- **#res_noise_constant=** Data Constant Error Term [V/I] (Res)
- **#ip_noise_percent=** Data Percent Errors [%] (IP)
- **#ip_noise_constant=** Data Constant Error Term [mV/V] (IP)
- **#use_robust_inv=** flag to enable Robust inversion algorithm (1 yes; 2 no)
- **#reweight_iterations_rho=** number to reweight iteration (Rho)
- **#reweight_iterations_ip=** number to reweight iteration (IP)
- **#plot_bounds=** array for bound values used during export to .vwer file
- **#save_iterations=** flag to save inverted model for each iteration (1 yes; 2 no)
- **#save_full_mesh=** flag to save both background and foreground (1 yes; 2 no)

```
#rough_trials_iter= 4 1 1 1 4 1 1 1 4 1 1 1 4 1 1
#constrain_to_starting= 1
#constrain_to_ref= 2
#max_num_iterations_rho= 3
#max_num_iterations_ip= 15
#max_internal_pcg_rho= 15
#max_internal_pcg_ip= 15
#inv_pcg_tolerance_rho= 0.001
#inv_pcg_tolerance_ip= 0.001
#optimal_rough_factor_rho= 0.9
#optimal_rough_factor_ip= 0.9
#rough_multiplier_rho= 10
#rough_multiplier_ip= 10
#initial_rough_rho= 10
#initial_rough_ip= 10
#x_rough_weight= 1
#y_rough_weight= 1
#z_rough_weight= 0.1

#res_noise_percent= 1
#res_noise_constant= 0.0001
#ip_noise_percent= 5
#ip_noise_constant= 1e-005
#use_robust_inv= 1
```

(continues on next page)

(continued from previous page)

```
#reweight_iterations_rho= 3
#reweight_iterations_ip= 3

#plot_bounds= 0 10 0 10 -5 0
#save_iterations= 1
#save_full_mesh= 1
```