GeoSLAM Volumes

Software Guide

Instructions for software configuration and operation

Notes

Revision: 1.3.1
Release date: December 2021.

GeoSLAM Ltd. provides this manual "as is", without warranty of any kind and reserves the right to make improvements and/or changes to the product described in this manual at any time and without notice.

SOFTWARE IS PROVIDED AS IS. To the maximum extent permitted by law, the manufacturer (GeoSLAM Ltd.) and its suppliers disclaim all warranties, either express or implied, including, but not limited to implied warranties of merchantability and fitness for a particular purpose, with regard to the SOFTWARE, the accompanying written materials, and any accompanying hardware. You may have other rights which vary from country to country.

NO LIABILITY FOR CONSEQUENTIAL DAMAGES. To the maximum extent permitted by applicable law, in no event shall the company or its suppliers be liable for any damages whatsoever (including without limitation, direct or indirect damages for personal injury, loss of business profits, business interruption, loss of business information, or any other pecuniary loss) arising out of the use of this product, even if the company has been advised of the possibility of such damages. In any case, the company's and its suppliers' entire liability shall be limited to the amount actually paid by the owner of the product for the product. Because some states/jurisdictions do not allow the exclusion or limitation of liability for consequential or incidental damages, the above limitation may not apply to the user.

Notation

	Information that supplements or clarifies text.
	A caution that actions, operation or configuration may lead to incorrect or improper use of the hardware/software.
	A warning that actions, operation, or configuration may result in regulatory noncompliance, safety issues or equipment damage.
Italics	Italics are used for unique terminology and part numbers, and for emphasis.
Options_-> Data	This notation is used for user interface controls and navigation.
`/data/dd_mm_yy_xxx`	This font is used for file and directory names, terminal interface input/output, and output in the text section of the main user interface window.

Revision History

Revision	Date	Description	Prepared by	Reviewed by
0.1	07/19	Working draft.	NH	JB
1.0	19/07/19	First revision.	NH	JB
1.1	21/08/19	Update to include features released in Version 1.1.	NH	AB
1.2	29/10/2019	Update to include features released in Version 1.2.	NH	AT

1.3	05/12/2020	Update to include features released in Version 1.3.	NH	AT
1.3.1	18/01/2021	Minor Update to include date string import template.	NH	AT

Contents

List of Figures

List of Tables

Part 1- Overview

GeoSLAM VOLUMES is a bespoke software designed for indoor and outdoor stockpile management. *GeoSLAM VOLUMES* was developed in close partnership with surveyors and stockpile managers to provide a simple-to-use, reliable system that has the flexibility and performance to function in a wide range of indoor and outdoor environments.

GeoSLAM VOLUMES is a standalone software package created by GeoSLAM to calculate near real-time volumetric data of a given commodity. This manual focuses on the analysis of the data. For acquisition, please consult manuals for the specific GeoSLAM Handheld Laser Scanners (HLS) or Fixed Laser Scanners (FLS).

Part 2 - Software Installation of this guide covers installation and licensing of the software package. Workflow instructions are provided in **Part 3 - User Workflows** with Reference Topics including the Program Interface provided in **Part 4 – Program Interface.** Background information concerning the concepts used in the software are provided in **Part 5 – Technical Background**.

In the event of a problem that cannot be resolved using the information supplied in this guide, please contact GeoSLAM. You can also gain assistance through the support page on our website: *https://geoslam.com/support/*

For further assistance, contact the GeoSLAM Technical Support by telephone or email. Our Customer Support personnel will discuss your situation, determine the cause of the problem, and provide the appropriate technical assistance.

Contact GeoSLAM by any of the following methods:

- Phone: +44 (0) 1157 270740 (all countries).
- Phone: +1-833-444-7907 (US & Canada).
- Email: support@geoslam.com

Part 2 - Software Installation

GeoSLAM VOLUMES is a bespoke software designed for indoor and outdoor stockpile management primarily from LiDAR based systems supported by *GeoSLAM*.

GeoSLAM VOLUMES is protected using Flexera software-based copy protection and licensing. A single licence key is provided with each copy unless further seats or licenses are purchased. The licence key can either be activated directly via the internet or using a manual activation for offline computers.

A simple user interface is provided with *GeoSLAM* software to facilitate user interaction with their Flexera licence. The utility is integrated into the *GeoSLAM VOLUMES* application.

License Manager

When the software is opened, the program will interrogate the license manager to determine if a valid software license is available. When the software is run for the first time after installation or if the licensing key has expired, then the software will fail the license check and display the *License Check* dialog (Figure 1).

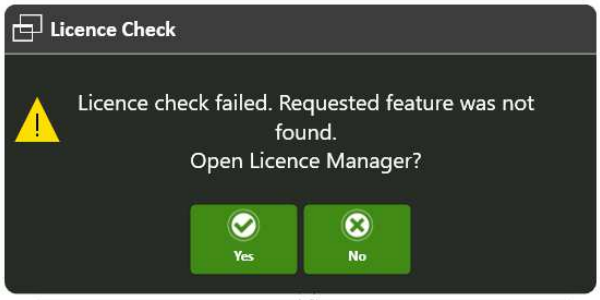


Figure 1. *License Check* dialog.

The license utility is accessible from the *File* menu bar in the *GeoSLAM Volumes* application. Selecting to open the license manager (Figure 2) will offer the user the choice to either activate the software **Online,** if the computer is connected to the internet, or **Offline** if the computer does not have an internet connection.

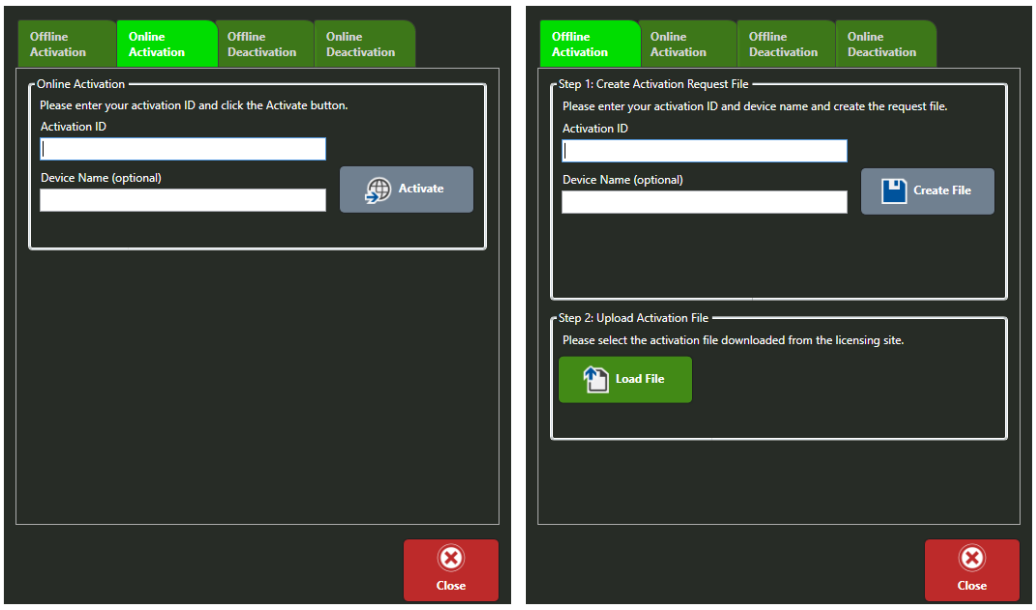


Figure 2. License Manager.

Additionally, the license manager offers the user the choice to either deactivate the software **Online,** if the computer is connected to the internet, or **Offline** if the computer does not have an internet connection.

Activation

Online Activation

This process is only applicable for computers with an internet connection. This connection can be a temporary connection in order to communicate with the licensing database. Once, the software is activated the software does not need to maintain this connection and the internet connection can be disabled.

To complete the Online Activation:

1. *Select* -> **Online Activation** tab (Figure 3).
2. *Input* -> **Activation ID** -> the activation code provided with the system.
3. The **Device Name** field can be left *Blank*.
4. *Select* -> **Activate**, to begin the activation process.

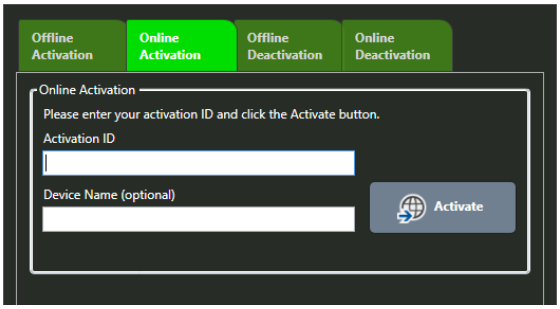


Figure 3. *Online activation* tab.

Ensure there are no spaces before or after your code. This is a common error made by the users that causes the activation to fail.

5. If a valid key has been entered, the successful *Information* dialog is displayed (Figure 4).

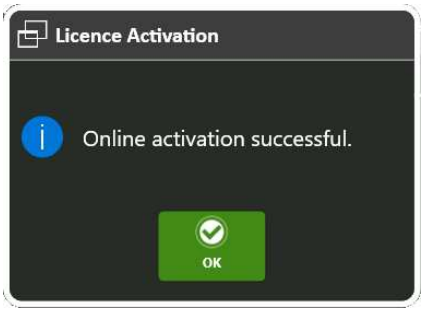


Figure 4. Successful activation dialog.

6. *Select* -> ***OK***, to close the dialog and continue.
7. *Select* -> **License Manager** -> **Close**, to continue.
8. This will display the **License Check** dialog (Figure 5).

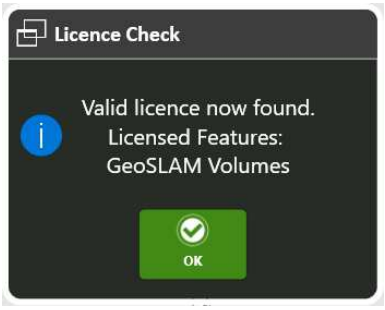


Figure 5. Valid license activation.

9. *Select* -> **OK** to continue. The software will automatically open.

Offline Activation

In order to undertake an offline activation, the user must have access to both the PC running the *GeoSLAM Volumes* software and a second PC that must have an internet connection.

To activate the license, the user must create a *Capability Request* file. This file must be then manually transferred from the *offline* PC to an internet enabled PC, which will then create a *Capability Response* using a web-based portal. The Capability Response file must then be transferred back from the internet enabled PC to the *offline* PC.

1. *Select* -> **Offline Activation** tab (Figure 6).
2. *Input* -> **Activation ID** -> the activation code provided with the system.
3. The **Device Name** field can be left *Blank*.
4. *Select* -> **Create File**, to create the *Capability Request* file. This will open the Create License Activation Request File dialog (Figure 7).
5. *Browse* -> save location.
6. *Select* -> **Save**, to continue.
7. The process will create 2 files (Figure 8). The *Capability Request* file plus a text file containing a link to the Flexera Network Operations (FNO) website.
8. Manually, copy the *Capability Request* file from the offline PC to the PC with internet connection.

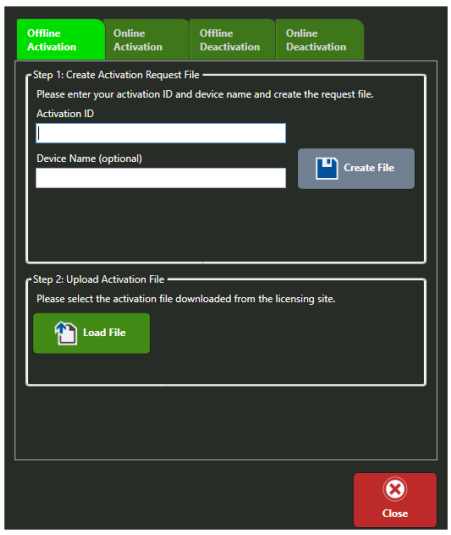


Figure 6. Online activation dialog.

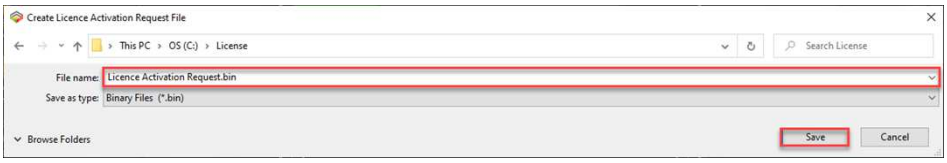


Figure 7. Create License Activation Request File dialog.

Figure 8. Files created during the Capability File request process.

9. From an internet enabled computer, open an internet browser.
10. Copy and paste the *FNO* login screen address from the text file to display the Login dialog (Figure 9).
11. At the log-in screen, *Select* -> **Activation ID**.

12. *Input* -> **Activation ID**, provided with the system.
13. *Select* -> **Log in**, to continue.

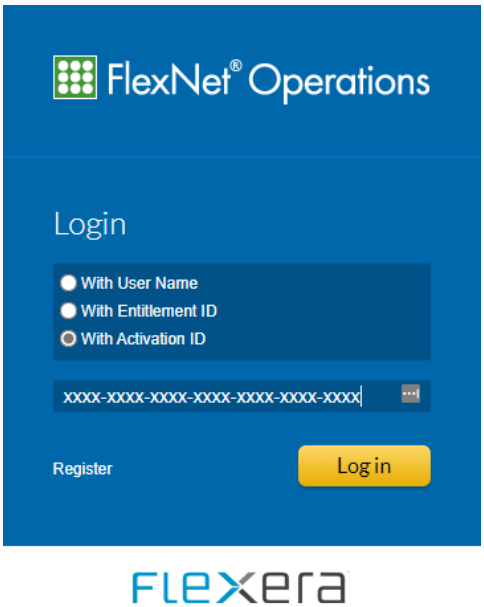


Figure 9. FNO login dialog from the website.

14. If the login process is successful, this will open the License and Delivery Portal (Figure 10).
15. *Select* -> **Devices** -> **Offline Device Management** (Figure 11).
16. This will open the **Offline Data Management** screen (Figure 12).
17. *Select* -> **Generate License.**
18. *Select* -> **Choose File** and browse to the location of the *Capability Request* file.
19. *Select* -> **Upload** to continue.
20. The system will offer the user the ability to download a *Capability Response* file.

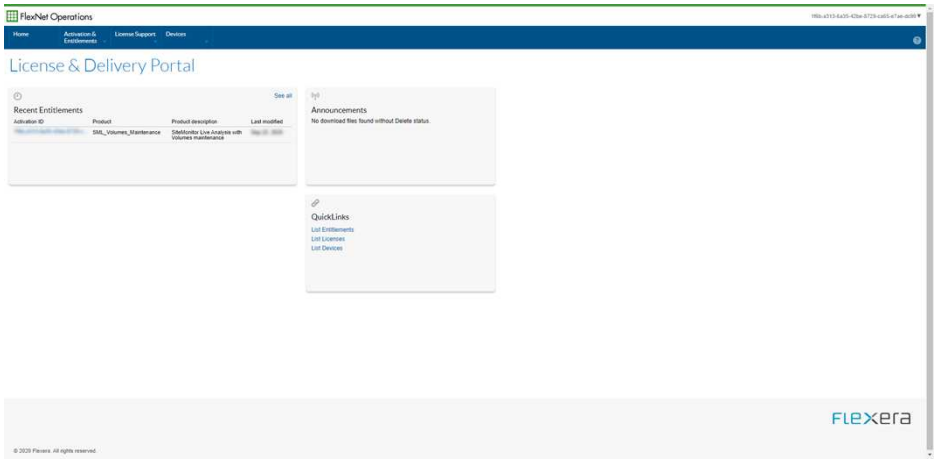


Figure 10. License and Delivery Portal.

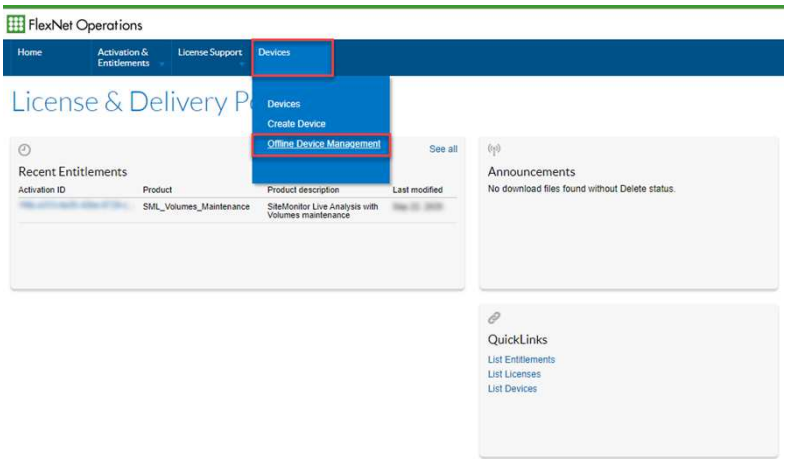


Figure 11. Select the Offline Device Management Option.

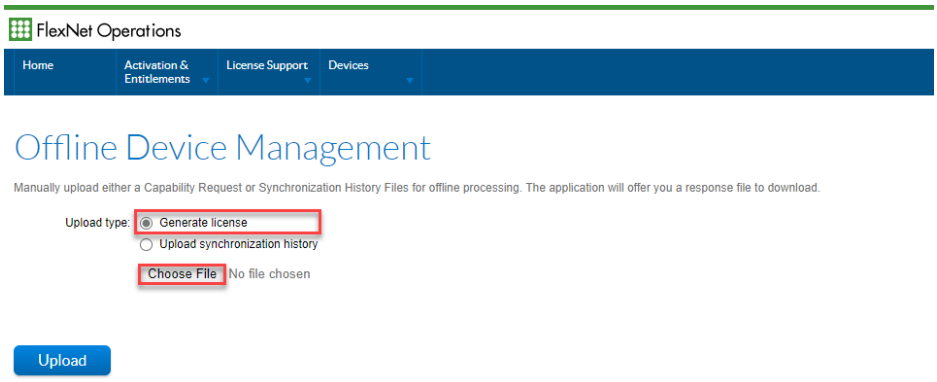


Figure 12. Offline Device Management.

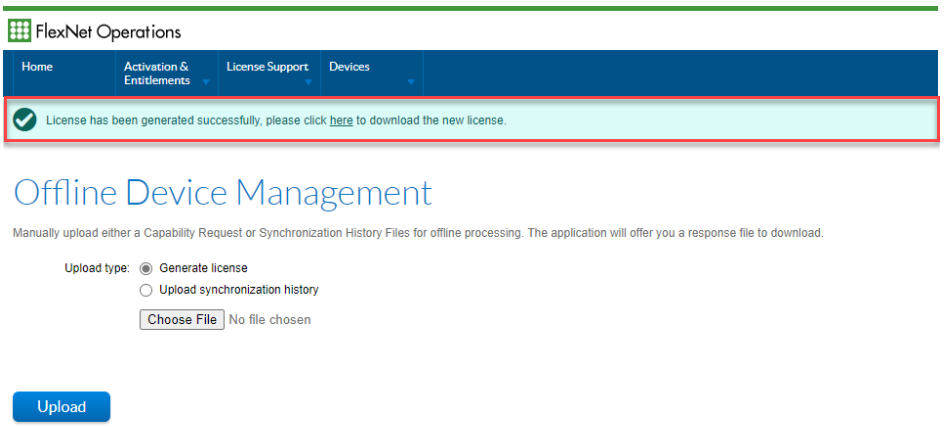


Figure 13. The FNO offline Device Management screen, with a link for downloading the *Capability Response* file.

21. Manually copy the *Capability Response* file back to the offline PC and return to the Offline Activation dialog.
22. *Step 2* -> *Select* -> **Load File** and browse to the location of the *Capability Response* file.
23. *Select* -> **Open,** to continue.
24. The software should now be activated and open to the start screen.

Deactivation

Online Deactivation

This process is only applicable for PC's with an internet connection. This connection can be a temporary connection in order to communicate with the licensing database. Once, the software is deactivated the software does not need to maintain this connection and the internet connection can be disabled.

To complete the online deactivation:

1. *Select* -> **Online Deactivation** tab (Figure 14).
2. *Input* -> **Activation ID** -> the activation code provided with the system.
3. The **Device Name** field can be left *Blank*.
4. *Select* -> **Deactivate**, to begin the activation process.

Ensure there are no spaces before or after your code. This is a common error made by the users that causes the activation to fail.

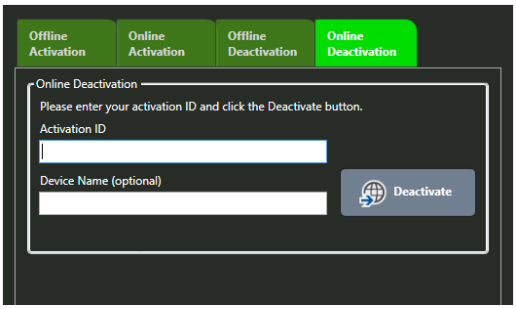


Figure 14. *Online deactivation* tab.

Offline Deactivation

In order to undertake an offline deactivation, the user must have access to both the PC running the *Volumes* software and a second PC that must have an internet connection.

To deactivate the license, the user must create a *Licence Deactivation Request* file. This file must be then manually transferred from the *offline* PC to an internet enabled PC, which then will create a *Capability Response* using a web-based portal. The Capability Response file must then be transferred back from the internet enabled PC to the *offline* PC. The software will then

create a *License Deactivation Confirmation* file that must be uploaded using the web-based portal.

1. *Select* -> **Offline Deactivation** tab (Figure 15).
2. *Input* -> **Activation ID** -> the activation code provided with the system.
3. The **Device Name** field can be left *Blank*.
4. *Select* -> **Create File**, to create the *Licence Deactivation Request* file. This will open the Create License Deactivation Request File dialog (Figure 15).
5. *Browse* -> save location.
6. *Select* -> **Save**, to continue.
7. The process will create 2 files (Figure 16). The *Licence Deactivation Request* file plus a text file containing a link to the Flexera Network Operations (FNO) website.
8. Manually, copy the *Licence Deactivation Request* file from the offline PC to the PC with internet connection.

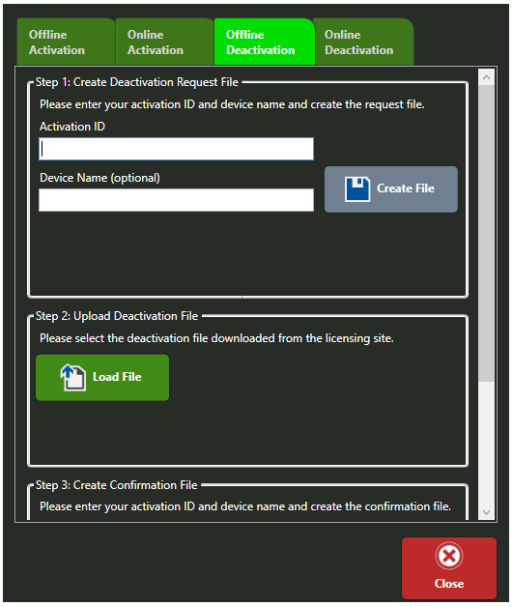


Figure 15. *Online deactivation* dialog.

Figure 16. Create License Deactivation Request File dialog.

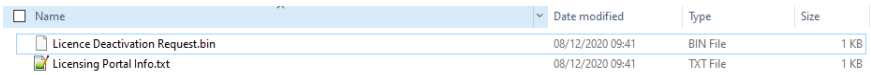


Figure 17. Files created during the Capability File request process.

9. From an internet enabled computer, open an internet browser.
10. Copy and paste the *FNO* Login screen address from the text file to display the Login dialog (Figure 18).
11. At the login screen, *Select* -> **Activation ID**.
12. *Input* -> **Activation ID**, provided with the system.
13. *Select* -> **Log in**, to continue.

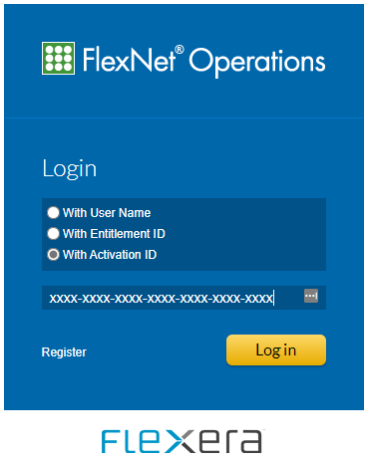


Figure 18. FNO login dialog from the website.

14. If the login process is successful, this will open the License and Delivery Portal (Figure 19).
15. *Select* -> **Devices** -> **Offline Device Management** (Figure 20).
16. This will open the **Offline Data Management** screen (Figure 21).
17. *Select* -> **Generate License.**
18. *Select* -> **Choose File** and browse to the location of the *Licence Deactivation Request* file.
19. *Select* -> **Upload** to continue.
20. The system will offer the user the ability to download a *Capability Response* file (Figure 22).

After downloading the *Capability Response* file, do not close the browser as the Portal will be required to confirm the deactivation.

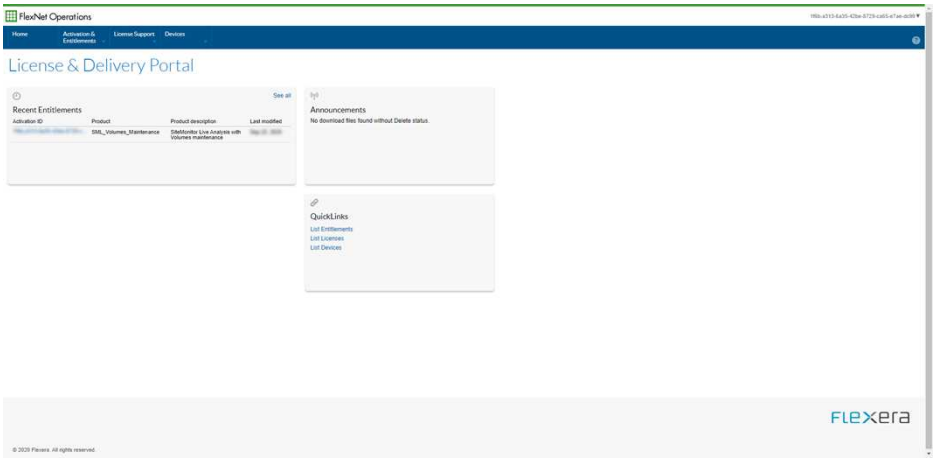


Figure 19. License and Delivery Portal.

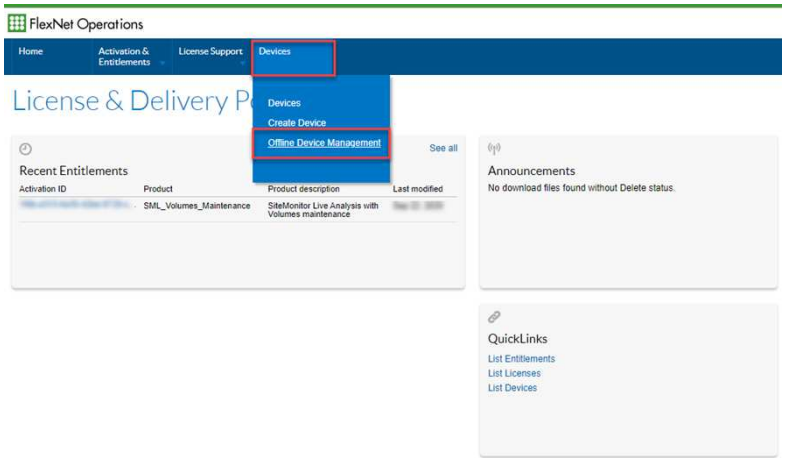


Figure 20. Select the Offline Device Management Option.

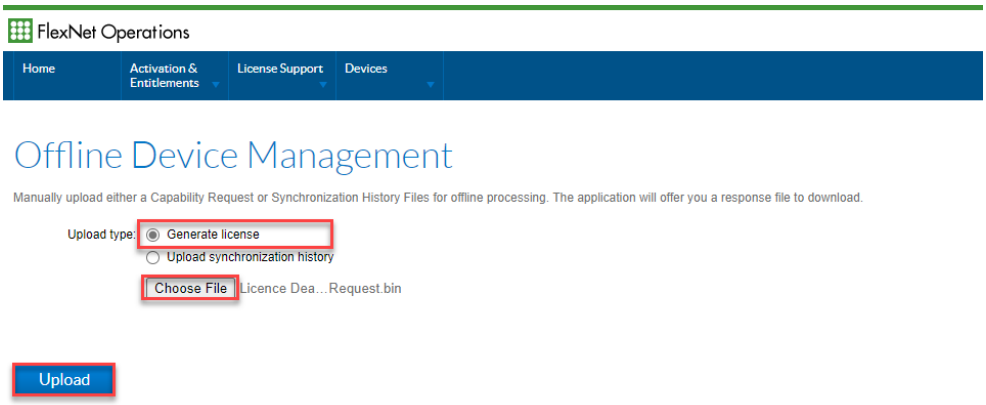


Figure 21. Offline Device Management to upload the *License Deactivation request* file.

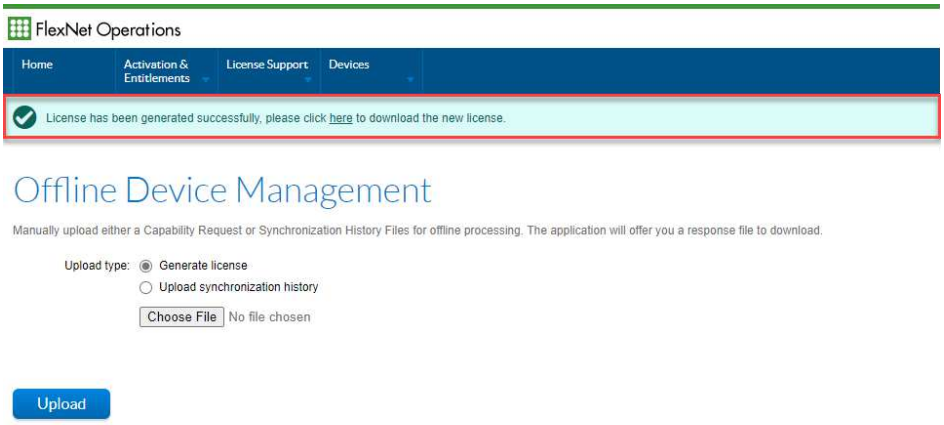


Figure 22. The FNO offline Device Management screen, with a link for downloading the "Capability Response" file.

21. Manually copy the *Capability Response* file back to the offline PC and return to the Offline Activation dialog.
22. *Step 2* -> *Select* -> **Load File** and browse to the location of the *Capability Response* file.
23. Select -> **Open** to continue.
24. This will open the *License Deactivation* confirmation dialog (Figure 23).

Figure 23. *Licence Deactivation* confirmation dialog.

25. *Select* -> **Yes** to confirm the deactivation.
26. This will create a *License Deactivation Confirmation* file.
27. *Browse* -> save location.
28. *Select* -> **Save** to continue.
29. *Return to the FNO web portal.*

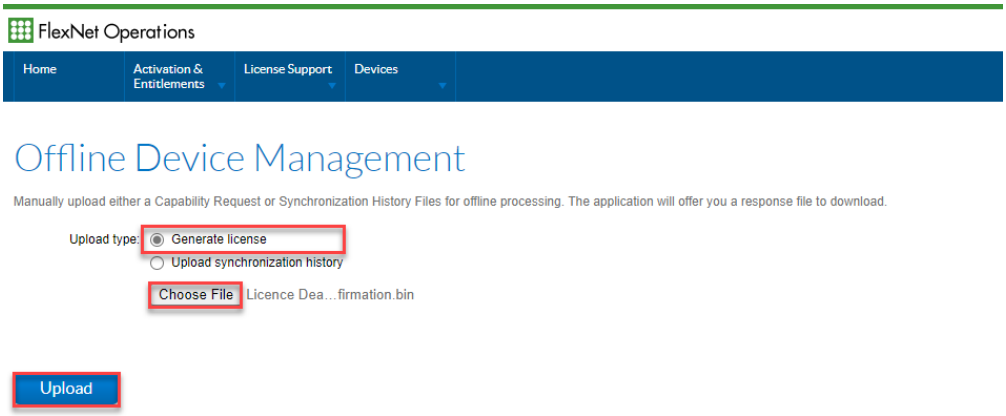


Figure 24. The FNO offline Device Management screen, with a link for uploading the *License Deactivation Confirmation* file.

30. At the **Offline Data Management** screen (Figure 24).
    a. *Select* -> **Generate License.**
    b. *Select* -> **Choose File** and browse to the location of the *License Deactivation Confirmation* file.
    c. *Select* -> **Upload** to continue.
31. *Close* -> **Browser**.
32. *Select* -> **Close**, at the *GeoSLAM Volumes* License Manager.
33. This will open the Licence check dialog to confirm that the licence is no longer valid (Figure 25).

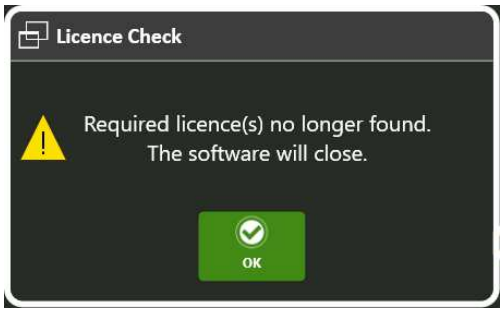


Figure 25. *License Check* dialog.

34. *Select* -> **OK** to continue.
35. The software will automatically close.

Upgrading the Software

Periodically, software updates will be provided by GeoSLAM that include new features, improvements, and resolution of issues in the software.

There is no need for the user to deactivate their current license when performing a software update.

To update the software:

1. Uninstall the current version of the software using the Windows *Apps and features* utility.
2. Install the new version of the software by running the installation executable and follow the on-screen instructions.

It is essential that before proceeding with any software update that the user contact GeoSLAM support to confirm that they have a valid maintenance contract in place.

Failure to ensure a valid maintenance contract may result in the software becoming non-functional.

Part 3 - User Workflows

This section describes the typical workflow when using *GeoSLAM Volumes*. The workflow is shown in Figure 26.

The basic workflow for projects is to create a new project; load a sample dataset and then define the floor, exclusion, and stockpile boundaries; set the material properties and refine any of the advanced processing parameters, e.g., file date string format.

Once project setup is complete, the sample dataset can be removed, and the user can choose to either import data manually or use the automated import function. New data is processed and visualised in both tabular and graphical format. Reports can be produced and if required, email alerts configured to trigger if pre-defined volumetric, tonnage or elevation levels are met.

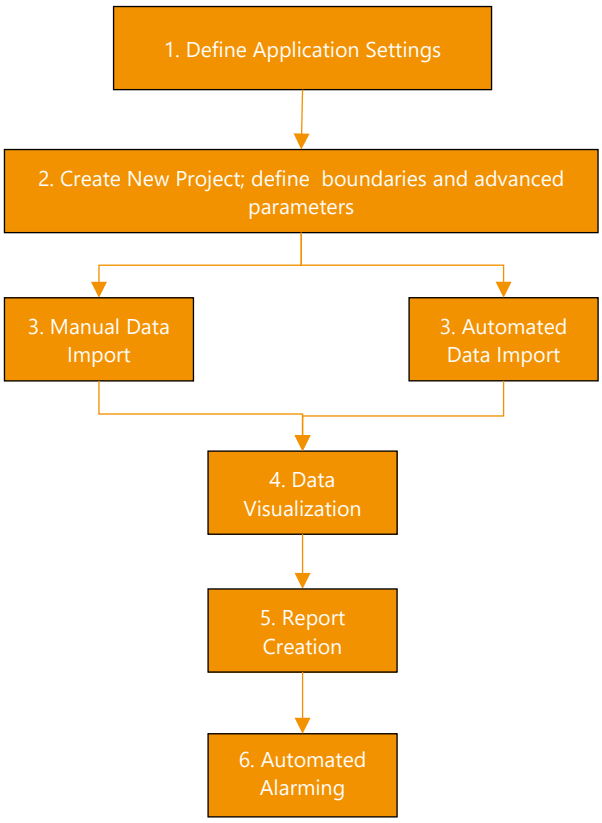


Figure 26. *GeoSLAM Volumes* user workflow.

Step 1. Define Application Settings

When the software is opened for the first time, the user must define the default application settings:

- Application Units (Metric, Imperial, US Survey).
- Reporting Logo.
- Date settings to be used to define the capture time and date.

Figure 27. Step 1: Workflow to define application settings.

1. Open *GeoSLAM Volumes*.
2. To set the application settings, *Select* -> **File** -> **Application Settings**.
    a. To set the global units:
        i. *Select* -> **Units** tab -> **Global Units** -> **System Units**.
        ii. Use the dropdown menu to select the required measurement system.
        iii. *Select* -> **Apply**.
        iv. *Select* -> **Save Application Settings** -> **OK.**
    b. To set the report logo:
        i. *Select* -> **Reporting** tab.
        ii. *Select* -> > **Browse**.
            1. Navigate to the system folder containing the required image. Most standard graphical formats are supported e.g., *.png format.
            2. *Select* -> **Open**.
        iii. *Select* -> **Apply**.
        iv. *Select* -> **Save Application Settings** -> **OK**.
    c. *To set the Default Date Settings*:
        i. *Select* -> **Enable Date from Scan Filename** -> **CHECKED**.
        ii. *Input* -> **Format** -> *user defined input string* following the format described in the *Date Format Help* section of the dialog.
        iii. *Select* -> **Apply**.
    d. *Select* -> **OK** to close the application settings dialog and continue.

Step 2 – Setting Up a New Project

When creating a new project, the user defines the project boundary (exclusion boundary), stockpile boundary, including material properties, and finally defines any advanced processing parameters that may be required. In order to set up the project, it is advisable that a sample dataset is loaded. A workflow for project creation is shown in Figure 28.

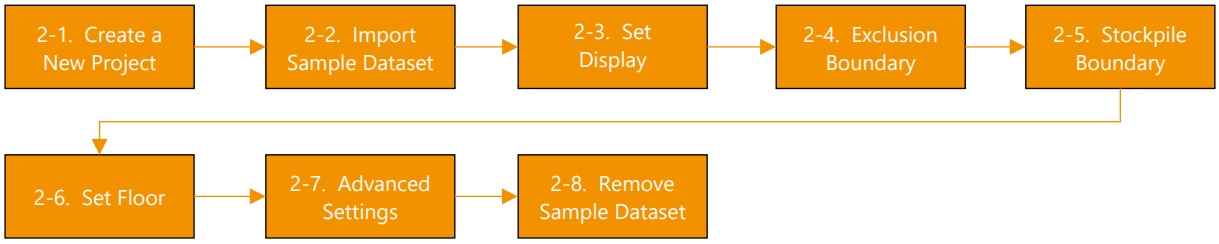


Figure 28. Step 2: Workflow to define new project.

Step 2-1. Creating and Configuring a New Project

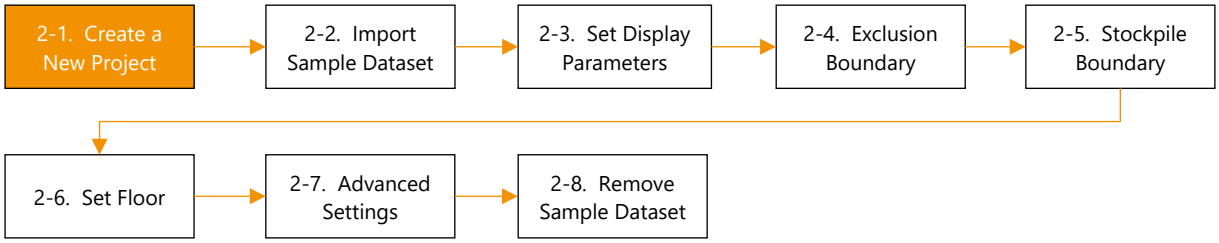


Figure 29. Step 2-1: New project creation workflow.

To create a new project,

1. *Select* -> **File** -> **New Project**.
    a. At the *Create New Volumes Project* dialog:
        i. *Input* -> **Project Name**.
        ii. *Input* -> **Directory** -> **Browse**.
            1. *Browse* -> *user defined project location.*
            2. *Select* -> **OK**, to continue.
        iii. *Select* -> **Create**, to continue.
    b. *Select* -> **Validate Project** -> **Yes**.

Step 2-2. Import a Sample Dataset

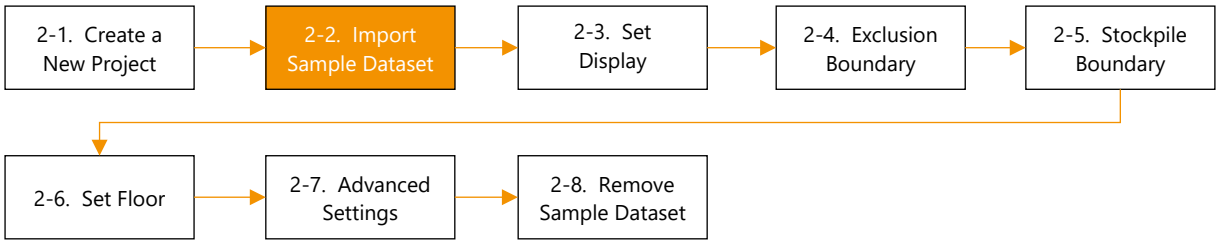


Figure 30. Step 2-2: Import a sample dataset workflow.

To import a sample dataset into the project:

1. *Select* -> **Scan Data** tab -> **Import** icon.
    a. *Browse* -> *Raw data folder*.
    b. *Select* –> *user defined data file* (.txt, .csv, .las, .laz).
    c. *Select* -> **Open**, to continue.
    d. *Select* -> **Data Import** -> **Process imported data** -> **No**.

   e. *Select* -> **Data Import** -> **OK**, to continue.
2. View the imported data.
   a. *Select* -> **Project Tree** -> *user data file*.
   b. *Select*-> *user data file* -> **Ticked**.

Step 2-3. Set Display Parameters

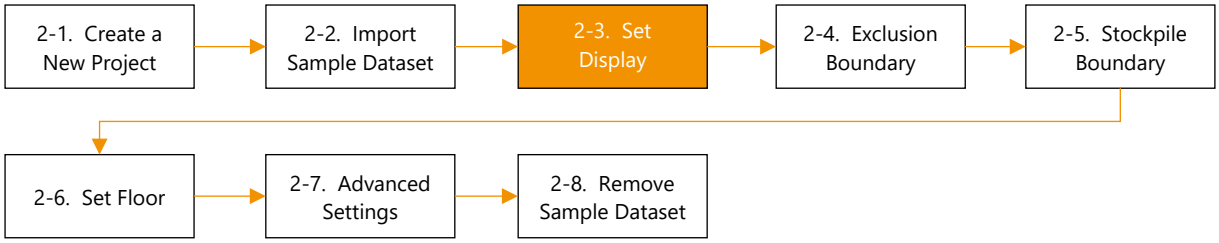


Figure 31. Step 2-3: Set display parameters workflow.

To set the display parameters:

1. Set the perspective:
    a. *Select* -> **Viewer** tab -> **Set View Mode** icon, to switch between the 2 display modes.
2. Set the Height Colouration:
    a. *Select* -> **Viewer** tab -> **Height Colouration** icon, to open the **Height Colouration** dialog.
    b. *Check* -> **Height Ramp**, to enable the colour ramp shader.
    c. The user can then manually set the **Max Elevation** (coloured Red) and **Min Elevation** (coloured Blue) by either using:
        i. *Select* -> **Slider** or *Input* -> *user defined value* to manually set the values, or
        ii. *Select* -> **Reset**, to automatically set the max and min values.
    d. *Check / Uncheck* -> **Show points outside elevation limits**, to either:
        i. display any points within the loaded dataset that are outside of the current shader values (these points will be displayed in Grey), or
        ii. Hide any points within the loaded dataset that are outside of the current shader values.
    e. *Select* -> **Close**, *to continue.*

Step 2-4. Set Exclusion Boundary

Exclusion Boundaries are created in the software to remove unwanted points outside of the area of interest.

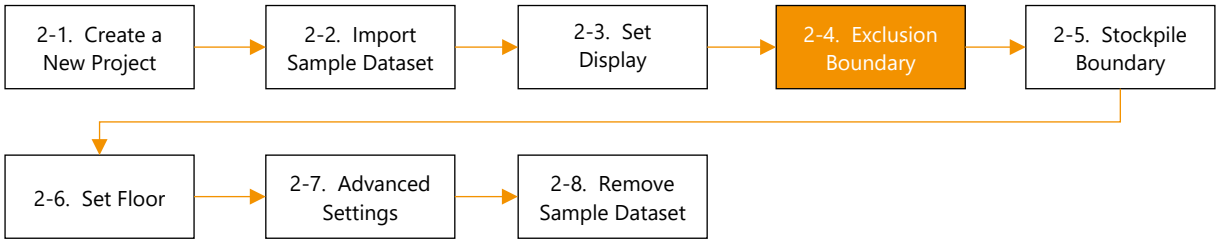


Figure 32. Step 2-4: Set exclusion boundary workflow.

There are several methods for Exclusion Boundary creation which can be implemented.

1. Using the Exclusion Boundary creation tool:
    a. Create Exclusion Boundary (rectangle), or
    b. Create Exclusion Boundary (polygon).
2. Loaded from a pre-existing file.

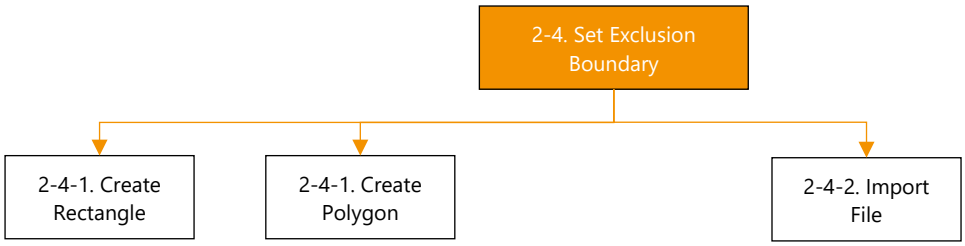


Figure 33. Step 2-4: Set exclusion boundary workflow subtasks.

To create a rectangular exclusion boundary:

1. *Select* -> **Editing** tab -> **Boundaries** icon.
    a. *Select* -> **Select Exclusion Boundary (Rectangle)**
        i. *Click* -> Top Left Corner of the rectangle.
        ii. *Hold and Drag* -> rectangle to the desired size. The boundary will be highlighted in Red.
    b. *Select* ->
        i. **Editing** tab -> **Apply**, to create the newly defined exclusion boundary into the project, or
        ii. **Editing** tab -> **Cancel**, to cancel the creation.

To create a polygon exclusion boundary:

1. *Select* -> **Editing** tab -> **Boundaries** icon.
    a. *Select* -> **Exclusion Boundary Select (Polygon)**.
        i. *Click* -> around the area to be defined. The area will be highlighted using a *Red* line.
        ii. *Double*-Click -> to close the area.
    b. *Select* ->
        i. **Editing** tab -> **Apply**, to create the newly defined exclusion boundary into the project, or
        ii. **Editing** tab -> **Cancel**, to cancel the creation.

<u>The newly created exclusion boundary will be visible in the *Exclusion Boundaries* category in the project tree.</u>

To import an Exclusion Boundary file.

1. *Select* -> **Editing** tab -> **Boundaries** icon.
    a. *Select* -> **Import Exclusion Boundary**.
    b. *Browse* -> *data folder*.
    c. *Select –> user defined exclusion boundary file*.
    d. *Select* -> **Open**, to continue.
    e. *Select* -> Boundary names, to import.
    f. *Select* -> **OK**, to import.
    g. *Select* -> **OK**, to continue.

<u>The newly imported exclusion boundary will be visible in the *Exclusion Boundary* category in the project tree.</u>

Having set up the exclusion boundary, ensure that it is visible and active in the project.

1. Ensure that the boundary is visible:
    a. *Select* -> **Project Tree** -> **Exclusion Boundary** -> *boundary name* -> **Ticked**.
2. Ensure that the Exclusion Boundary is active:
    a. *Select* -> **Project Tree** -> **Exclusion Boundary** -> *boundary name* -> *Boolean Slider* -> **Ticked**.

Step 2-5. Set Stockpile Boundary

Stockpile boundaries are created in the software to enable specific areas of interest to be identified for volumetric analysis. Boundary setup involves:

1. Creating / importing a stockpile boundary.
2. Setting the material properties within the boundary.

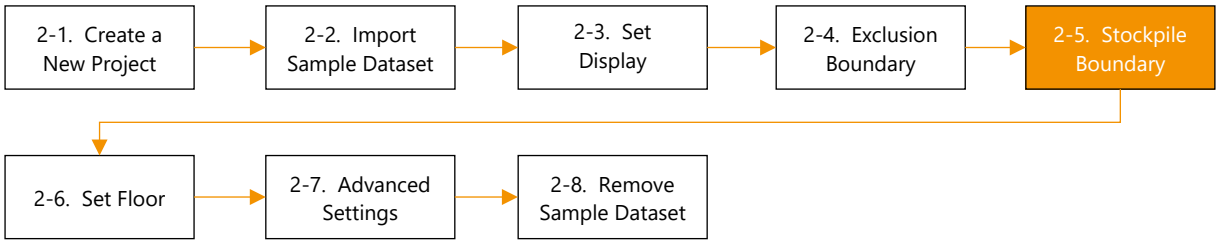


Figure 34. Step 2-5: Set stockpile boundary workflow.

There are several methods for Stockpile Boundary creation which can be implemented.

1. Using the Stockpile Boundary creation tool:
    a. Create Stockpile Boundary (rectangle), or
    b. Create Stockpile Boundary (polygon).

2. Loaded from a pre-existing file.

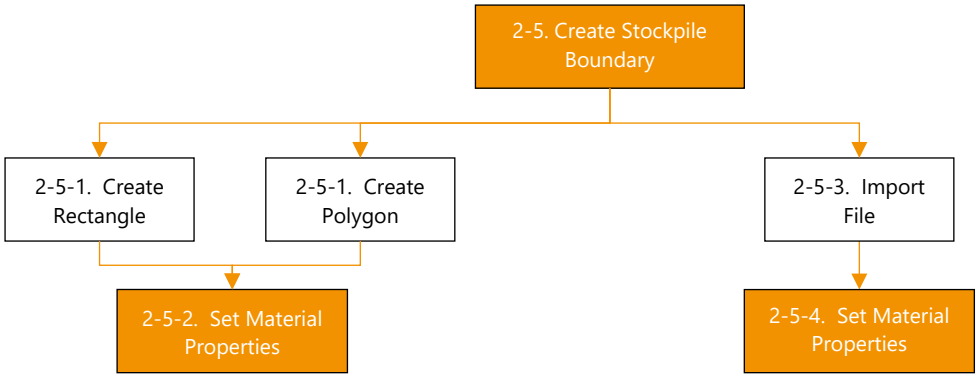


Figure 35. Step 2-5: Set stockpile boundary workflow subtasks.

To create a rectangular stockpile boundary:

1. *Select* -> **Editing** tab -> **Boundaries** icon.
    a. *Select* -> **Select Stockpile Boundary (Rectangle)**.
        i. *Click* -> Top Left Corner of the rectangle.
        ii. *Hold and Drag* -> rectangle to the desired size. The boundary will be highlighted in Green.
    b. *Select* ->
        i. **Editing** tab -> **Apply**, to create the newly defined stockpile boundary into the project, or
        ii. **Editing** tab -> **Cancel**, to cancel the creation.
2. Selecting **Apply** will open the *Create Boundary* dialog where the stockpile material properties are set. From the Create Boundary dialog:
    a. *Input* -> **Name** -> user defined name for the boundary.
    b. *Input* -> **Specific Gravity** of the material within the bounds of the area.
    c. *Input* -> **Cut Bulking Factor** of the material within the bounds of the area.
    d. *Input* -> **Fill Bulking Factor** of the material within the bounds of the area.
    e. *Select* -> **Save** to create the boundary or **Cancel** to return.

To create a polygon stockpile boundary:

1. *Select* -> **Editing** tab -> **Boundaries** icon.
    a. *Select* -> **Stockpile Boundary Select (Polygon)**.
        i. *Click* -> around the area to be defined. The area will be highlighted using a *Green* line.
        ii. *Double*-Click -> to close the area.
    b. *Select* ->
        i. **Editing** tab -> **Apply**, to create the newly defined stockpile boundary into the project, or

ii. **Editing** tab -> **Cancel**, to cancel the creation.

2. Selecting **Apply** will open the *Create Boundary* dialog where the stockpile material properties are set. From the Create Boundary dialog:
    a. *Input* -> **Name** -> user defined name for the boundary.
    b. *Input* -> **Specific Gravity** of the material within the bounds of the area.
    c. *Input* -> **Cut Bulking Factor** of the material within the bounds of the area.
    d. *Input* -> **Fill Bulking Factor** of the material within the bounds of the area.
    e. *Select* -> **Save**, to create the boundary or **Cancel** to return.

<u>The newly created stockpile boundary will be visible in the *Boundaries* category in the project tree.</u>

To import a stockpile boundary from a file:

1. *Select* -> **Editing** tab -> **Boundaries** icon.
    a. *Select* -> **Import Boundary**.
        i. *Browse* -> *data folder*.
        ii. *Select* –> *user defined boundary file*.
        iii. *Select* -> **Open**, to continue.
        iv. *Select* -> Boundary names to import.
        v. *Select* -> **OK**, to import.
        vi. *Select* -> **OK**, to continue.
2. Set the material properties for the boundaries loaded from the file:
    a. *Select* -> **Settings** -> **Material Properties** icon.
        vii. *Highlight* -> **Boundary Name**.
            1. *Input* -> **Specific Gravity**.
            2. *Input* -> **Cut Bulking Factor**.
            3. *Input* -> **Fill Bulking Factor**.
            4. *Select* -> **Save**, to continue.

Having set up the Stockpile Boundary, ensure that it is visible and active in the project.

1. Ensure that the boundary is visible:
    a. *Select* -> **Project Tree** -> **Boundaries** -> *boundary name* -> **Ticked**.
2. Ensure that the boundary is active:
    a. *Select* -> **Project Tree** -> **Boundaries** -> *boundary name* -> *Boolean Slider* -> **Ticked.**

Further stockpile boundaries can be added by repeating these steps.

Step 2-6. Set Floor

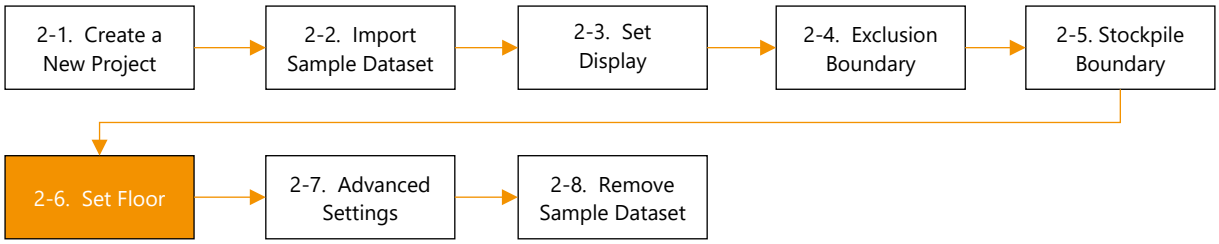


Figure 36. Step 2-6: Set floor workflow.

There are several types of floor which can be implemented.

1. Flat planar floor which can be either:
    a. Horizontal at a fixed height, or
    b. An inclined plane defined by selecting 3 points in the dataset.
2. Pre-existing reference data set.

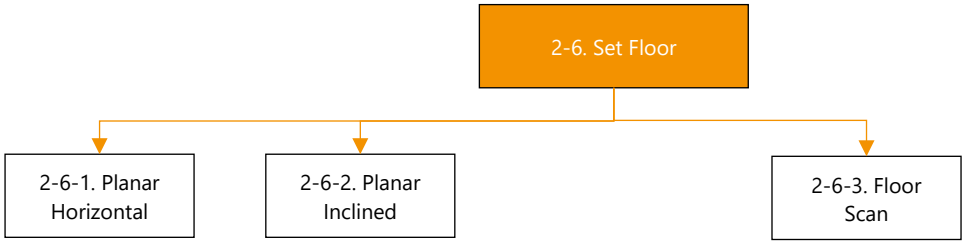


Figure 37. Step 2-6: Set floor workflow subtasks.

To create a planar reference floor at a fixed height:

1. *Select* -> **Editing** Tab -> **Set Floor** icon.
2. *Select* -> **Set Planar Floor**, to open the *Set Floor* dialog.
    a. *Select* -> **Planar Floor -> Reset**, to choose the lowest elevation value, or
    b. Select -> **Planar Floor** -> **Elevation** -> Use the slider to set a user defined reference floor height, or
    c. Input -> **Planar Floor** -> **Elevation** -> user defined height into the input box.
3. *Select* -> **Close**, to continue.

To create an incline planar reference floor:

1. Ensure that a scan dataset is visible in the viewer.
2. *Select* -> **Editing** tab -> **Set Floor** icon.
3. *Select* -> Set **Planar Floor**, to open the *Set Floor* dialog.
4. *Select* -> **Planar Floor** -> **Select Plane** -> **Start**.
    a. *Shift* + *Click* -> 3 points in the dataset.

b. Each selected point will be identified by a marker.

5. With 3 points selected the inclined plane will be displayed.
6. *Select* -> **Close**, to continue.

To import a scan file:

1. *Select* -> **Editing** Tab -> **Set Floor** icon.
2. *Select* -> **Import Floor**.
    i. Input -> *Name.*
    ii. *Select* -> *Import File Path*
        1. *Browse* -> *Raw data folder*.
        2. *Select* –> *user defined floor file*.
        3. *Select* -> **Open**, to continue.
    iii. *Select* -> **Import**, to continue.

Having created the reference floor, ensure that the floor is visible and active in the project.

1. Ensure that the reference floor is visible:
    a. *Select* -> **Project Tree** -> *Planar floor* -> **Ticked**, or
    b. *Select* -> **Project Tree** -> *user floor file* -> **Ticked**.
2. Ensure that the reference floor is active:
    a. *Select* -> **Project Tree** -> *Planar floor* -> *Boolean Slider* -> **Ticked**, or
    b. *Select* -> **Project Tree** -> *user floor file* -> *Boolean Slider* -> **Ticked**.

Step 2-7. Set Advanced Settings

If the sample data contains noise or includes features that need to be removed (e.g., infrastructure) or contains excessive datapoints, the user can set a series of advanced processing parameters to decimate and/or clean the data.

By default, if the Default Date Settings has been configured at the application level, then new projects will have Date Settings enabled automatically. The date format entered in the application settings will be used at the project level. The user has the ability to turn off or set a new date format string at a project level in the Advanced Settings dialog.

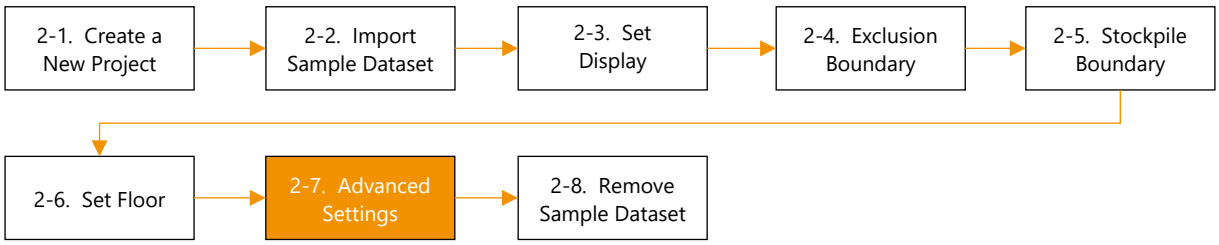


Figure 38. Step 2-7: Set advanced settings workflow.

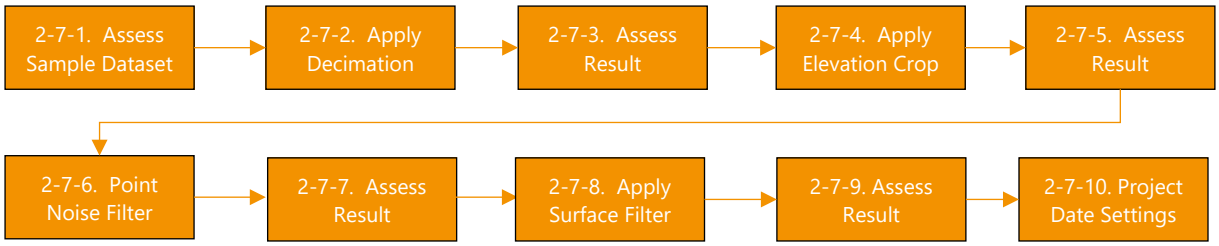


Figure 39. Step 2-7: Set advanced settings workflow subtasks.

1. Assess the dataset to identify if advanced processing is required.
2. If the sample data contains excessive data, then the pointcloud can be decimated to reduce processing times.
    a. *Right-Click* -> **Project Tree** -> **Scans** -> *user data file* -> **Remove**.
        i. *Select* -> **Yes**, at the confirmation dialog.
    b. Set the Advanced Settings:
        i. *Select* -> **Settings** tab -> **Advanced Settings** icon.
        ii. *Select* -> **Processing Settings** tab.
        iii. *Select* -> **Processing Settings** -> **Point Decimation** -> **CHECKED**.
        iv. *Input* -> **Minimum Point Separation** -> *user defined value.*
        v. *Select* -> **Grid Spacing** -> **Grid Size** -> *user defined value* e.g., grid size should be no less than ¼ of the point decimation.
        vi. *Select* -> **OK**, to continue.
        vii. *Select* -> **Save Settings** -> **OK**.
3. Visualise the sample data and assess the data quality.
    a. Import the sample dataset and do not process the data.
    b. Assess the effect of the settings.
    c. If necessary, repeat number Step 2 or continue to Step 4.

4. If the sample data contains points well above or well below the expected extents of the data, then an elevation crop can be applied.
    a. *Right-Click* -> **Project Tree** -> **Scans** -> *user data file* -> **Remove**.
        i. *Select* -> **Yes**, at the confirmation dialog.
    b. Set the Advanced Settings:
        i. *Select* -> **Settings** tab -> **Advanced Settings** icon.
        ii. *Select* -> **Processing Settings** tab.
        iii. *Select* -> **Processing Settings** -> **Elevation Cropping**-> **CHECKED**.
        iv. *Input* -> **Keep Elevation in Range** -> *user defined value to user defined value e.g.*, 2m below the lowest point expected in any dataset -> 1m above the highest point expected in any dataset.
        v. *Select* -> **OK**, to continue.
        vi. *Select* -> **Save Settings** -> **OK**.
5. Visualise the sample data and assess the data quality.
    a. Import the sample dataset and do not process the data.
    b. Assess the effect of the settings.
    c. If necessary, repeat number Step 4 or continue to Step 6.

6. If the sample data contains isolated points, a point noise filter can be applied.

a. *Right-Click* -> **Project Tree** -> **Scans** -> *user data file* -> **Remove**.
    i. *Select* -> **Yes**, at the confirmation dialog.
b. Set the Advanced Settings:
    i. *Select* -> **Settings** tab -> **Advanced Settings** icon.
    ii. *Select* -> **Processing Settings** tab.
    iii. *Select* -> **Processing Settings** -> **Point Noise Filter**-> **CHECKED**.
        1. *Input* -> **Number of Neighbouring Points** -> *user defined value to user defined value e.g., 6.*
        2. *Input* -> **Standard Deviation Threshold**-> *user defined value to user defined value e.g., 1.*
    iv. *Select* -> **OK**, to continue.
    v. *Select* -> **Save Settings** -> **OK**.

7. Visualise the sample data and assess the data quality.
    a. Import the sample dataset and do not process the data.
    b. Assess the effect of the settings.
    c. If necessary, repeat number Step 6 or continue to Step 8.

8. If the sample data contains noise points close to the ground, infrastructure that must be removed or moving items, then a surface filter can be applied to try and remove these features.
    a. *Right-Click* -> **Project Tree** -> **Scans** -> *user data file* -> **Remove**.
    b. Set the Advanced Settings:
        i. *Select* -> **Settings** tab -> **Advanced Settings** icon.
        ii. *Select* -> **Processing Settings** tab.
        iii. *Select* -> **Processing Settings** -> **Surface Filter**-> **CHECKED**.
            1. *Set* -> **Site Type** -> **Stockpile** or **Earthworks**.
            2. *Set* -> **Smoothing** -> **Checked** or **Unchecked**.
            3. *Input* -> **Sampling** -> *user defined value,* e.g., 0.3.
            4. *Input* -> **Max Iteration** -> *user defined value,* e.g., 500.
            5. *Input* -> **Classification Threshold** -> *user defined value*, e.g., 0.3.
            6. *Set* -> **Return Only Surface** -> **Checked** or **Unchecked,** e.g., typically checked.
        iv. *Select* -> **OK**, to continue.
        v. *Select* -> **Save Settings** -> **OK**.
9. Visualise the sample data and assess the data quality.
    a. Import the sample dataset and do not process the data.
    b. Visualise the effect of the settings.
    c. If necessary, report number Step 8 or Select -> **OK**, to close the dialog and continue.

10. To ensure that the correct data capture time is used for the tabular or graphical displays, it is advised that users enter the capture date and time into the filename to be loaded. The user defines the format of this date and time in the application settings. The user has the option to turn off (or on) or redefine a project specific date and time format.

    a. *Select* -> **Settings** tab -> **Advanced Settings** icon.

b. *Select* -> **Date Settings** tab.
c. *Select* -> **Project Date Settings** -> **Extract Date from Scan Filename** -> **CHECKED** or **UNCHECKED**.
    i. *If checked the user can choose to keep the default format or enter a new project specific format.*
d. *Select* -> **OK**, to continue.
e. *Select* -> **Save Settings** -> **OK**.

Step 8. Remove Sample Dataset

Having completed the basic tasks for a new project; created a new project; loaded a sample dataset and then defined the floor, exclusion, and stockpile boundaries; set the material properties and refined any of the advanced processing parameters, the project setup is complete. Any sample data should be removed before loading more data and running the complete processing workflow.

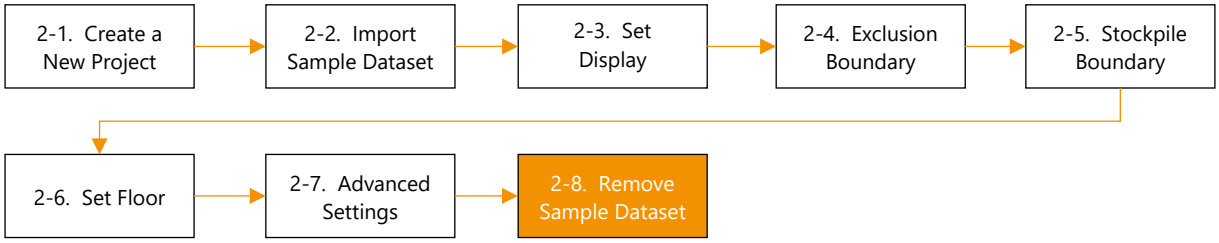


Figure 40. Step 2-8: Remove sample dataset workflow.

To remove any sample datasets from the project:

1. *Right-Click* -> **Project Tree** -> **Scans** -> *user data file* -> **Remove**.
2. *Select* -> **Yes**, at the confirmation dialog.

Step 3. Data Import

Data can be imported into the software using either:

1. Manual Import function.
2. Automatic Import function.

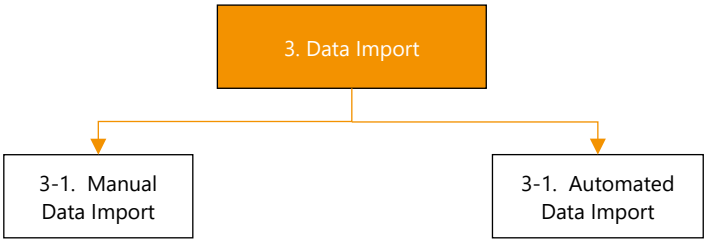


Figure 41. Step 3: Data import workflow.

To manually import a dataset into the project:

1. *Select* -> **Scan Data** tab -> **Import** icon.
    a. *Browse* -> *Raw data folder*.
    b. *Select* –> *user defined data file*.
    c. *Select* -> **Open**, to continue.
2. *Select* -> **Data Import** -> **Process imported Data** -> **Yes**.
3. View the imported data.
    a. *Select* -> **Project Tree** -> *user data file*.
    b. *Select*-> *user data file* -> **Ticked**.

To configure the automated processing:

1. *Select* -> **Scan Data** tab -> **Start Auto** icon.
    a. From the Set Data Folder dialog:
        i. *Select* -> **Scan Data Folder** -> **Browse**.
            1. *Navigate* -> raw data folder to use.
        ii. It is useful to minimise the number of scans held in the interface with older scans being archived. Data is not deleted but moved to an archive folder.
            1. *Select* -> **Enable Archiving** -> **CHECKED**.
            2. *Select* -> **Archive Data Folder** -> Browse.
                a. *Navigate* -> *user project* -> *ProcessData* folder.
                b. *Select* -> **Make New Folder** -> *Archive*.
                c. *Select* -> **OK**, to continue.
            3. *Input* -> **Archive After** -> *xxx* scans.
        iii. *Check* -> **Auto Resume** -> **CHECKED** (if required).
        iv. *Select* -> **Save**, to continue.
2. Any data that is copied into the **Scan Data Folder,** whilst the software is running, will be automatically processed.
3. View the imported data.
    a. *Select* -> **Project Tree** -> *user data file*.
    b. *Select*-> *user data file* -> **Ticked**.
4. Automated data loading can be stopped at any time.
    a. *Select* -> **Scan Data** tab -> **Stop Auto** icon.

Step 4. Data Visualisation

The user can visualise the processed volumetric and tonnage data via both the:

- Data Table.
- Data Graph.

For both the table and the graph the user has the ability to alter the date range for which data is displayed.

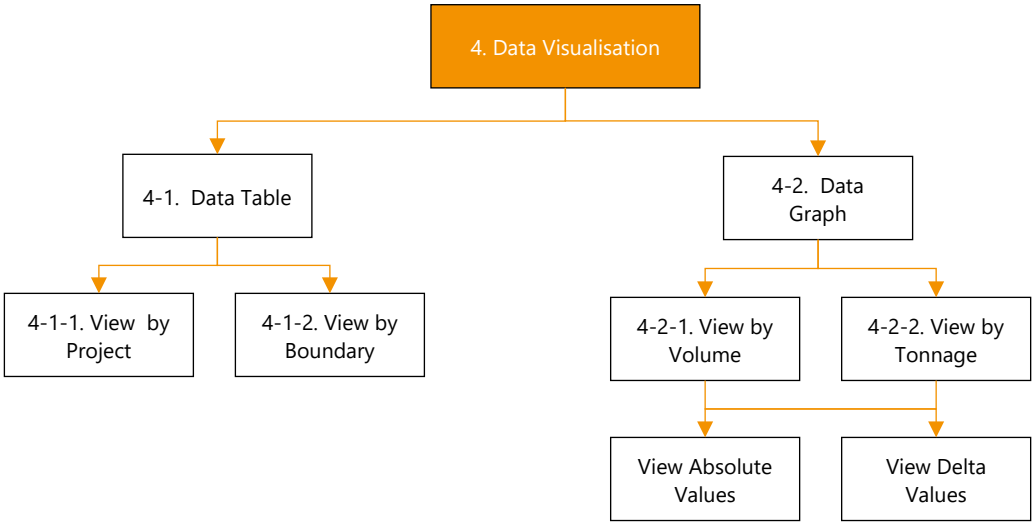


Figure 42. Step 4: Data visualisation workflow.

Step 4-1 Data Visualisation – Data Table

To view the processed data metrics for the global project in the data table:

1. *Select* -> **Data Table** tab.
    a. *Select* -> **Source** dropdown -> **Project**.
    b. To filter by date:
        i. *Select* -> **Calendar** icon, to choose the start date.
        ii. *Select* -> **Calendar** icon, to choose the end date or *Check* -> **Use Current Time**, to display data including the latest scan.
2. The data table will automatically update with the specified data.

To view the processed data metrics for an individual boundary:

1. *Select* -> **Data Table** tab.
    a. *Select* -> **Source** dropdown box -> **Boundary**.
    b. *Check* -> **Boundaries** dropdown box -> *Active Boundary* to display.
    c. *To filter by date,*
        i. *Select* -> **Calendar** icon to choose the start date.
        ii. *Select* -> **Calendar** icon to choose the end date or Check -> Current time to display the latest scans.
2. The Data Table will automatically update with the specified data.

Step 4-2 Data Visualisation – Graph

To view the data graphically in the Graph view:

1. *Select* -> **Graph** tab.
2. *Select* -> **Date Filters** ->
    a. *Select* -> either **Day**, **Week**, **Month**, or
    b. *Select* -> **Range** ->
        i. *Select* -> **Calendar** icon, to choose the start date.
        ii. *Select* -> **Calendar** icon, to choose the end date or Check -> Current time to display the latest scans.
3. *Select* -> **Settings** ->
    a. Volume or Tonnage. Only one selection can be made.
    b. Absolute or Delta. Only one selection can be made.
    c. Cut, Fill or Net. Multiple selections can be made.
4. *Select* -> **Boundaries** -> *boundary name* -> *Boolean* -> **Active**. Only one selection can be made.

Step 5. Report Creation

The *Generate Report* function allows the user to create a PDF report showing volumetric or tonnage data per boundary over a user defined timescale. A copy of the data table can be included in the report.

Figure 43. Step 5: Report creation workflow.

To create a report:

1. *Select* -> **Reporting** tab -> **Generate Report** icon.
2. This will open the *Generate Report* dialog.
    a. *Select* -> **Settings** -> **From** -> start date of report.
    b. *Select* -> **Settings** -> **To** -> start date of report.
    c. *Select* -> **Boundaries** -> *Check* -> *boundary names* to include in the report.
    d. *Select* -> **Measurements** -> *Check* -> *Cut*, *Fill* and / or *Net* values to report.
    e. *Select* -> **Measurement Type** -> *Volume or Tonnage* values to report.
    f. *Select* -> **Include data table** -> *Check* -> to include the data table in the report.

To preview the report:

1. *Select* -> **Preview**, to view a preview of the report. The report will include as a minimum:
    a. The reporting period.
    b. The boundaries reported.
    c. For each boundary, the following parameters are reported:
        i. Boundary Area.

ii. Specific Gravity.
iii. Cut Bulking Factor.
iv. Fill Bulking Factor.

d. Screenshot of the graph covering the report time period.

If report is satisfactory, save it to file.

1. *Select* -> **Save PDF**.
2. *Browse* -> user folder to save the report.
   a. Select -> **Save**, to continue.
3. Once the report has been created a *Generate Report* confirmation dialog will be displayed.
4. *Select* -> **OK**, to continue.
5. *Select* -> **Close**, to close the dialog and continue.

Step 6. Automated Alarming

With the alarming function, the user has the ability to create email alerts that are triggered against user defined levels. Triggers are set at the boundary level.

- Net Volumes.
  - Minimum Level.
  - Maximum Level.
- Net Tonnage.
  - Minimum Level.
  - Maximum Level.
- Height above Floor.
  - Minimum Level.
  - Maximum Level.

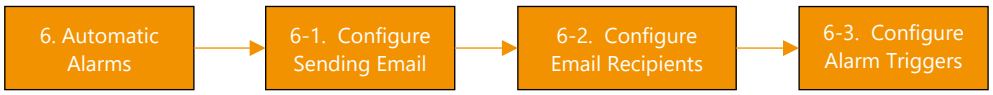


Figure 44. Step 6: Automated alarm creation workflow.

To configure the email alert settings:

1. Configure the email settings from where alarms are sent.
   a. *Select* -> **Settings** tab -> **Email Settings** icon.
   b. This will open the *Project Settings* dialog.
      i. *Input* -> **Username** -> *username for the email account.*
      ii. *Input* -> **Password** -> *password for the email account.*
      iii. *Input* -> **Host** -> *user specified email server*, e.g., smtp.office365.com.
      iv. *Input* -> **Port**-> *user specified port number.* Port 587 is the standard port assignment for SMTP email.

v. *Input* -> **Sending Email Address** -> The email address from which alerts are sent.
vi. *Enable* -> Use SSL. SSL (Secure Sockets Layer) is the standard security technology for establishing an encrypted link between a server and a client.

2. Configure the alarm email recipients.
    a. *Select* -> **Settings** tab -> **Alarm Setup** icon.
    b. This will open the *Alarm Setup* dialog.
        i. *Input* -> **Receiving Email Addresses** -> email addresses of the people to receive the email alerts.
        ii. *Select* -> *Boundary Name*.
            1. Input -> Trigger Values ->
                a. **Net Volumes** -> *Min* and / or *Max*, as required.
                b. **Net Tonnage** -> *Min* and / or *Max*, as required.
                c. **Height Above the Floor** -> *Min* and / or *Max*, as required.
        iii. *Select* -> **Save**, to continue.

Part 4 – Program Interface

GeoSLAM VOLUMES is a bespoke software designed for indoor and outdoor stockpile management from LiDAR based systems, particularly the Handheld and Fixed Laser Scanner systems supported by *GeoSLAM*.

File Formats

GeoSLAM VOLUMES can utilise data formatted in several common pointcloud data formats, allowing the software to analyse data from multiple acquisition sources.

Data type	Format	Format type
Input data format	CSV, TXT LAS, LAZ 3DP	XYZ Ascii Binary GeoSLAM Binary
Input boundary format	DXF	Drawing Exchange Format
Project settings	.vp	Binary
Exclusion Files	DXF	Drawing Exchange Format
Boundary Files	DXF	Drawing Exchange Format

Table 1. File formats used by *GeoSLAM Volumes*.

Program Window

Opening the program will display the main screen (Figure 45). The main window is divided into 2 areas:

1. Menu and toolbar containing the quick access tools.
2. Data analysis and visualisation window.

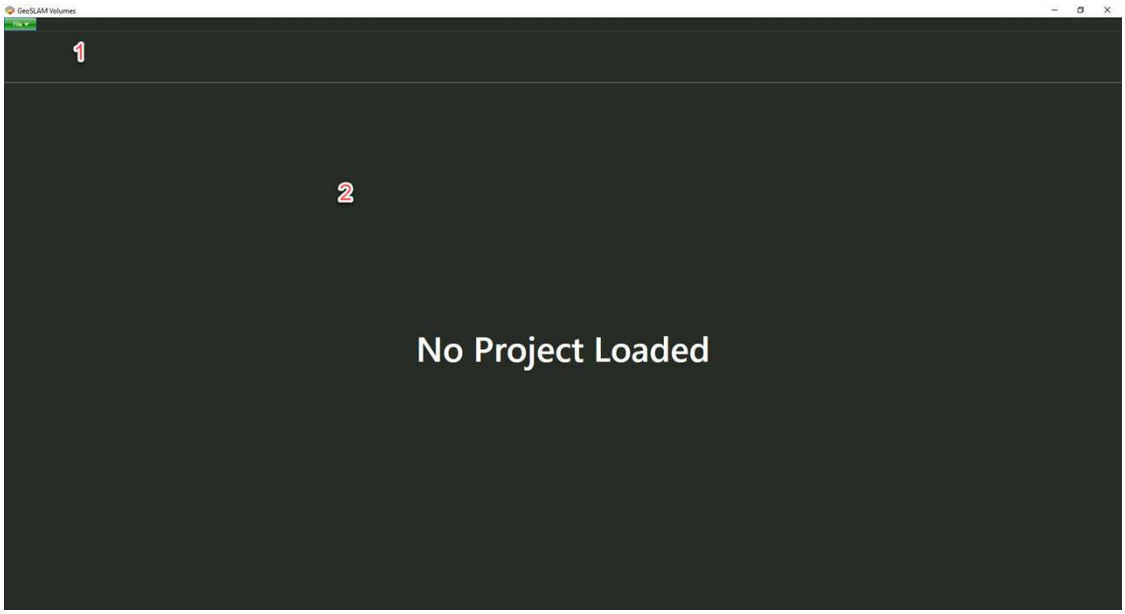


Figure 45. Main *GeoSLAM VOLUMES* screen with no project open.

Main Window Layout

Once a project is active, the main screen with all controls will be visible (Figure 46 and Figure 47). A description of the screen components highlighted in Figure 46 is provided in Table 2.

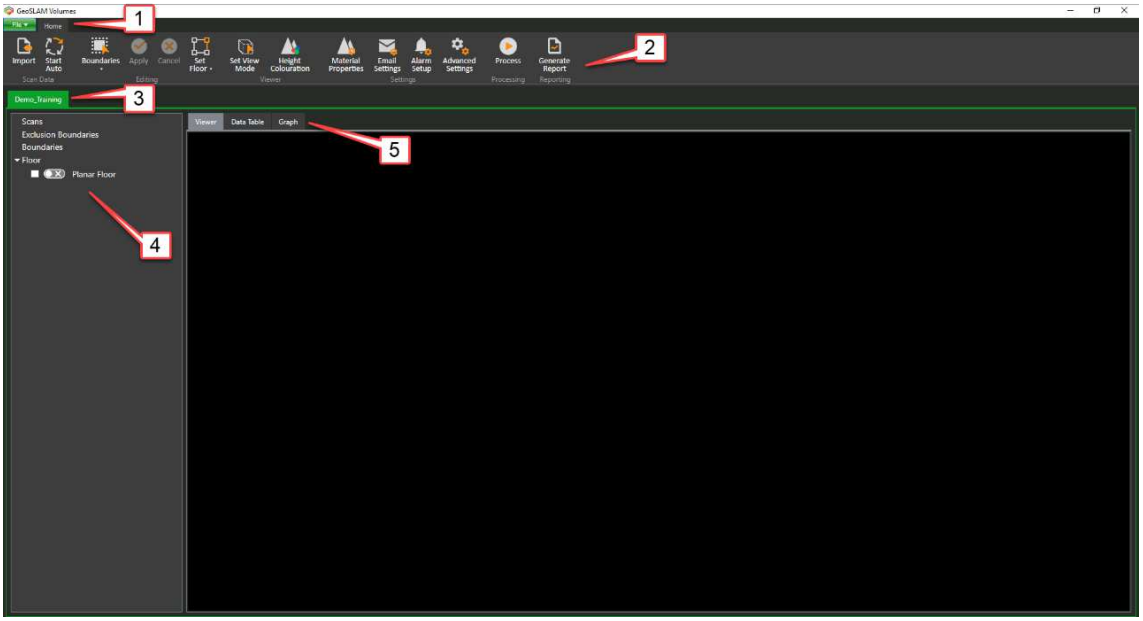


Figure 46. An open project with no data imported.

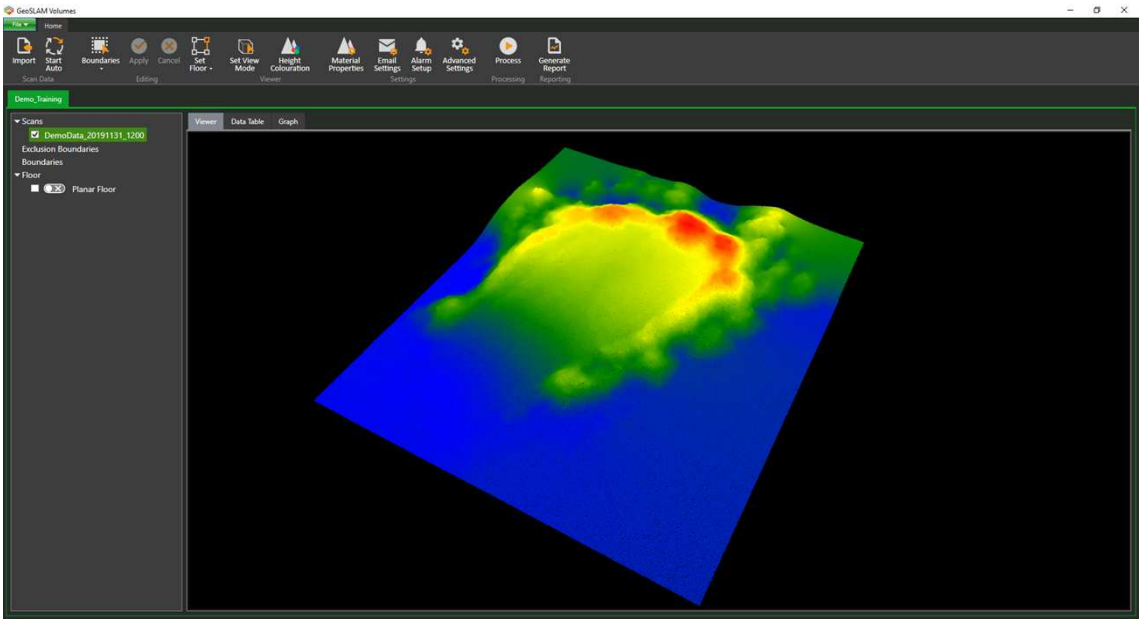


Figure 47. An open project with data imported.

Number	Item	Description
1	Menu bar	The menu bar contains access to the dropdown system menus.
2	Toolbar	This is the main user interface for setting up the project and interacting with the data.
3	Project Information Bar	Users have the ability to be able to load multiple projects into the software. Each project will have a separate tab on the project information bar.
4	Project Tree	All Scan, Boundary and Floor data loaded into the project is listed in the project tree. The user has the option to make an item visible, to make the item active or non-active as well as removal or export options.
5	Data Display	The data display window provides the user with 3 distinct options to interrogate the data. The user can select from: Viewer – display the raw pointcloud data. Data Table - display the processed volumetric data in tabular form. Graph - view a graphical representation of the data history.

Table 2. Features of the main window.

Detailed descriptions for each of the 5 panels of the main window are described below.

Panel 1 – Menu bar

The menu bar comprises:

- File Menu -> Contains options to either create a new analysis project or open an existing project, as well as giving access to global application settings and licensing options.
- Home Menu -> Contains the primary toolbar for interacting with the software.

File Menu

Selecting the File Menu provides the user with several options to either create a new analysis project or open an existing project, as well as define global application settings. The file menu also provides access to the License Manager.

Menu	Description
File -> New Project	Create a new *GeoSLAM Volumes* project.
File -> Open Project	Open an existing project.
Recent Projects	Lists the recently opened projects.
Licensing	Opens the licensing dialog.
Application Settings	Opens the applications settings dialog.
Exit	Closes *GeoSLAM Volumes*.

Table 3. Menu Options.

Creating a New Project

To create a new analysis project:

1. Open *GeoSLAM Volumes*.
2. *Select* -> **File** -> **New Project**.
    a. At the *Create New Volumes Project* dialog (Figure 48).
        i. *Input* -> **Project Name**.
        ii. *Input* -> **Directory** -> **Browse.**
            1. *Browse -> user defined project location.*
            2. *Select* -> **OK**, to continue.

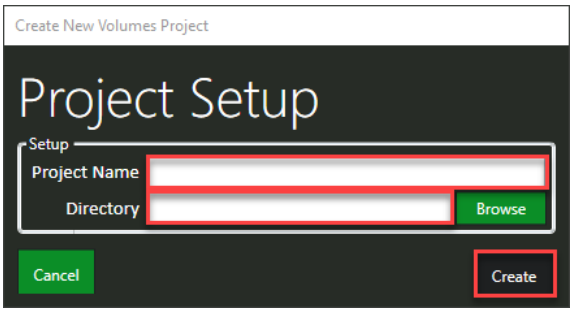


Figure 48. New Project Creation Dialog.

The options within the *Create New Volumes Project* dialog are described in Table 4.

Option	Description
Project name	The user must enter a name to define the project.
Directory	Use the *Browse* icon to navigate to the folder in which the project directory will be created.

Table 4. New Project Creation Dialog Options.

If the user enters a project name that already exists, or the directory is invalid then the *Create* icon will remain inoperable.

3. *Select* -> **Create**, to open the *Validate project* dialog (Figure 49).
4. *Select* -> **Yes**, to continue.

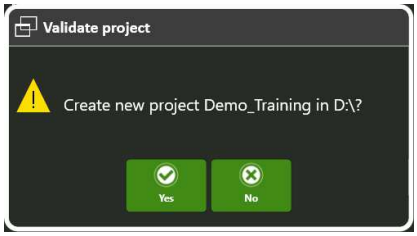


Figure 49. Validate new project dialog.

Opening an existing project

To open an existing analysis project:

1. Open -> *GeoSLAM Volumes*.
2. *Select* -> **File** -> **Open Project,** to open the navigation dialog.
3. *Browse* -> Directory containing the project file and select the project *.vp file.
4. *Select* -> **Open**, to continue.

Recent

The *File Menu* displays the recently opened analysis projects.

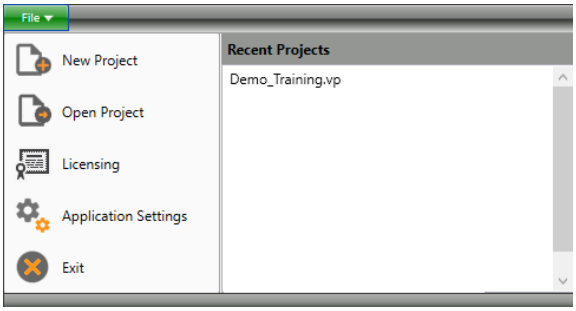


Figure 50. Recent Projects.

To open a recent project:

1. *Select* -> project name (*.vp) from the list to open the project.

Only one instance of a project can be opened at the same time.

Licensing

Selecting the licensing option will open the *License Manager* (Figure 51) allowing the user to activate a new license key or deactivate an existing license key. Details on using the *License Manager* are provided in Part 2 - Software Installation.

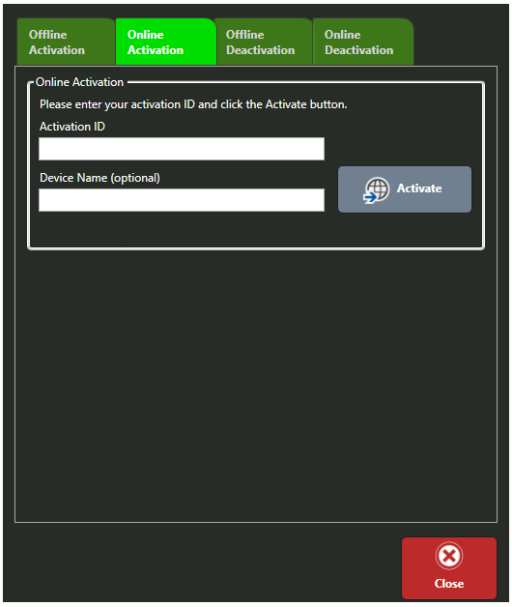


Figure 51. *GeoSLAM License Manager*.

Application Settings

The *Application Settings* dialog (Figure 52) provides the user with the ability to define global application settings. The user has the option to:

1. Set system units.
2. Set a logo that will be used in the reporting feature.
3. Define Date settings that will be used to extract date and time from the filename of an input file.

Figure 52. Application Settings dialog.

Units

The user can set the unit measurement system used by the software. Unit measurements systems supported are:

1. Metric.
2. Imperial.
3. US Survey.

Setting the global system unit will affect the material properties definition, the processed volumetric and tonnage reporting units and applicable advanced processing parameter units.

System Units	Parameter			
	Area	Volume	Tonnage	Bulk Density
Metric	m^2	m^3	Metric Tonnes	kg/m^3
Imperial	yd^2	yd^3	Long Tonnes	lbs/yd^3
US Survey	yd^2	yd^3	Short Tonnes	lbs/yd^3

Table 5. Parameter units defined by the global system units.

The user can alter the Measurement Units at any time. All parameters will be automatically updated within the user interface.

To set the measurement units:

1. *Select* -> **File -> Application Settings**.
2. By default, this will display the *Units* tab (Figure 53).
3. *Select* -> **Global Units** -> **System Units**. Use the dropdown menu to select the required measurement system.

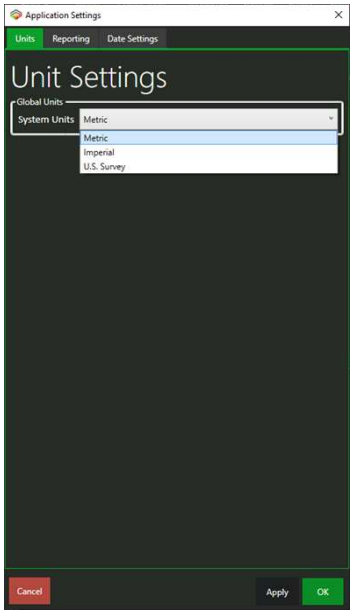


Figure 53. Unit Settings tab.

4. *Select* -> **Apply**.
5. *Select* -> **OK**, to confirm that the application settings have been successfully saved.
6. *Select* -> **OK**, to close the dialog and continue.

Reporting

When using the reporting function within the software, the user has the option to include an image, e.g., company logo, that will be automatically added to the report header.

To set the report logo:

1. *Select* -> **File Applications**.
2. This will open the Application Settings dialog (Figure 53).
3. *Select* -> **Reporting** tab (Figure 54).

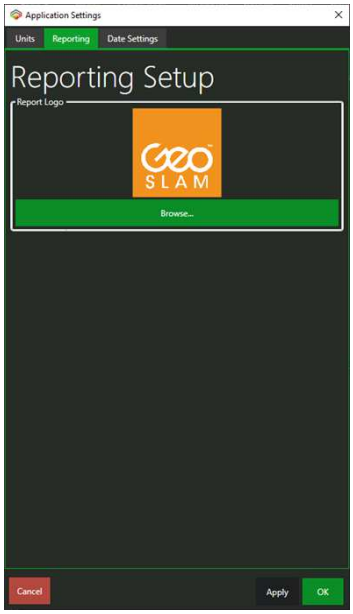


Figure 54. Reporting Setup tab.

4. *Select* -> **Report Logo** -> **Browse**.
5. Navigate to the system folder containing the required image. Most standard graphical formats are supported e.g., *.png format.
6. *Select* -> **Open**.
7. *Select* -> **Apply**.
8. *Select* -> **OK**, to confirm that the application settings have been successfully saved.
9. *Select* -> **OK**, to close the dialog and continue.

Default Date Settings

Precise date and time information is essential for accurate reporting and graphing of volumetric data. By default, the software will assign the date and time of a scan from the *modified* date in the data file. However, the date and time stored in the *modified* file attribute does not always reflect the time at which the file is first created. Copying and Pasting files between computers via a USB drive, or transferring data over FTP can alter the *modified* date / time attribute. When the data is then imported into the software interface, this will cause anomalous results to be observed in the tabular and graphing windows.

To overcome this issue, it is recommended that users always store the date and time of a scan in the filename. By creating a date setting filename template, the software will extract this information and use it for tabulation, graphing and reporting.

To set a date setting template:

1. *Select* -> **File Applications**.
2. This will open the Application Settings dialog (Figure 53).
3. *Select* -> **Date Settings** tab (Figure 55).

Figure 55. Reporting Setup tab.

4. *Select* -> **Extract Date From Scan Filename** -> **CHECKED**.
5. *Input* -> **Format** -> *user defined format*.
    a. It is recommended to use filenames with the date string at the end of the filename, e.g., *stockpile1_16012021_1325.csv*.
    b. In this case the user would enter -> **ddMMyyyy_HHmm*

Details of the Date Formats that can be used to define the template are given in Table 6.

6. *Select* -> **Apply**.
7. *Select* -> **OK**, to confirm that the application settings have been successfully saved.
8. *Select* -> **OK**, to close the dialog and continue.

Date Format	Description
d	Day of the month, from 1 through 31.
dd	Day of the month, from 01 to 31.
ddd	Abbreviated day of the week, e.g., "Mon".
dddd	Full day of the week, e.g., "Monday".
M	Month, from 1 through 12.
MM	Month, from 01 through 12.
MMM	Abbreviated name of the month, e.g., "Jun".
MMMM	Full name of the month, e.g., "June".
y	Year, from 0 to 99.
yy	Year, from 00 to 99.
yyy	Year with a minimum of 3 digits.
yyyy	Year as a four-digit number.
h	Hour in 12-hour format, from 1 through 12.
hh	Hour in 12-hour format, from 01 through 12.
H	Hour in 24-hour format, from 0 through 23.
HH	Hour in 24-hour format, from 00 through 23.
m	Minute, from 0 through 59..
mm	Minute, from 00 through 59.
s	Second, from 0 through 59.
ss	Second, from 00 through 59.
t	First character of the AM/PM designator, e.g., "A"
tt	AM/PM designator, e.g., "AM"
":" "/" "-" " " "," "_"	Separators.
"*"	Wildcard. One wildcard is allowed at the beginning or end of the format string. Allows for text to be ignored at the beginning of end of the filename.

Table 6. Default Data Settings template options.

Panel 2 - Toolbar

The toolbar panel is the main user interface for setting up the project and interacting with the data. Toolbar Functions are described in Table 7.

Icon	Tab	Function	Description
	Scan Data	Import	Opens a secondary dialog enabling the user to browse to the location of data for manual import.
		Start Auto	Opens a secondary dialog where the user defines the location of the data folder to monitor for new data for automated processing. Also, sets a folder for data archiving.
	Editing	Boundaries	Opens the boundary definition tool that allows the user to define Exclusion and Stockpile boundaries. These can either be a) rectangular, b) polygonal or c) a pre-defined boundary file (in DXF format).
		Apply	Confirm a boundary definition and add it into the project.
		Cancel	Cancel a boundary definition.
		Set Floor	Opens secondary dialogs enabling the user to either a) set a planar floor or b) load a pre-existing surface file. The planar floor option provides the user to either create a horizontal floor or create an inclined plane by selecting 3 points in the data set. Each of these 3 points will act to define the plane.
	Viewer	Set View Mode	Allows the user to choose between viewing the 3D data in either an Orthometric or Perspective camera mode.
		Height Colouration	Allows the user to set the colour ramp by which data is coloured.
	Settings	Material Properties	To enable tonnage to be calculated the user must define the material properties of Specific Gravity and Cut and Fill Bulking Factor. These parameters are defined as either a project setting or per boundary.
		Email Settings	Opens a secondary dialog to define the username and email host details from where email alerts are sent.
		Alarm Setup	Opens a secondary dialog to define the recipients of any alarm emails plus which boundaries are monitored.
		Advanced Settings	Allows the user to define the processing options for the project. Processing options are defined on data import.
	Processing	Process	Allows the user to manually start data processing. Manual processing can be triggered independent of where manual or automatic import is enabled.
	Reporting	Generate Report	Allows the user to create a user definable report (PDF format) to summarise the volumetric data.

Table 7. Toolbar Functions.

Scan Data Tab

GeoSLAM Volumes can process data in two distinct modes.

- Manual.
- Automatic.

In manual mode, the user must copy data into a folder and then browse to the data using the *Import Data* icon.

In automatic mode, the user selects a specific folder that is monitored by the software. When any new data file is identified in the folder, it is automatically imported and volumetric analysis undertaken. In this mode, scans can be automatically archived from the interface for more efficient memory management.

Import

To manually import pointcloud data:

1. *Select* -> **Scan Data** tab -> **Import** icon.
2. From the *Open* dialog, browse to the data folder containing the data to import.
3. *Select* -> *user defined file*.
4. *Select* -> **Open**.
5. This will open the *Data Import* dialog (Figure 56).
    a. *Select* -> **Yes**, to process the data immediately.
    b. *Select* -> **No**, to simply import the data and use the manual process function after viewing the data.
6. *Select* -> **OK**, at the confirmation dialog to continue.

Figure 56. Dialog to choose to process the manually imported data.

When loading the first data set into a new project, the user is advised NOT to process data until the rest of the project setup is complete.

The new data will be visible in the *Scans* category in the project tree (Figure 57).

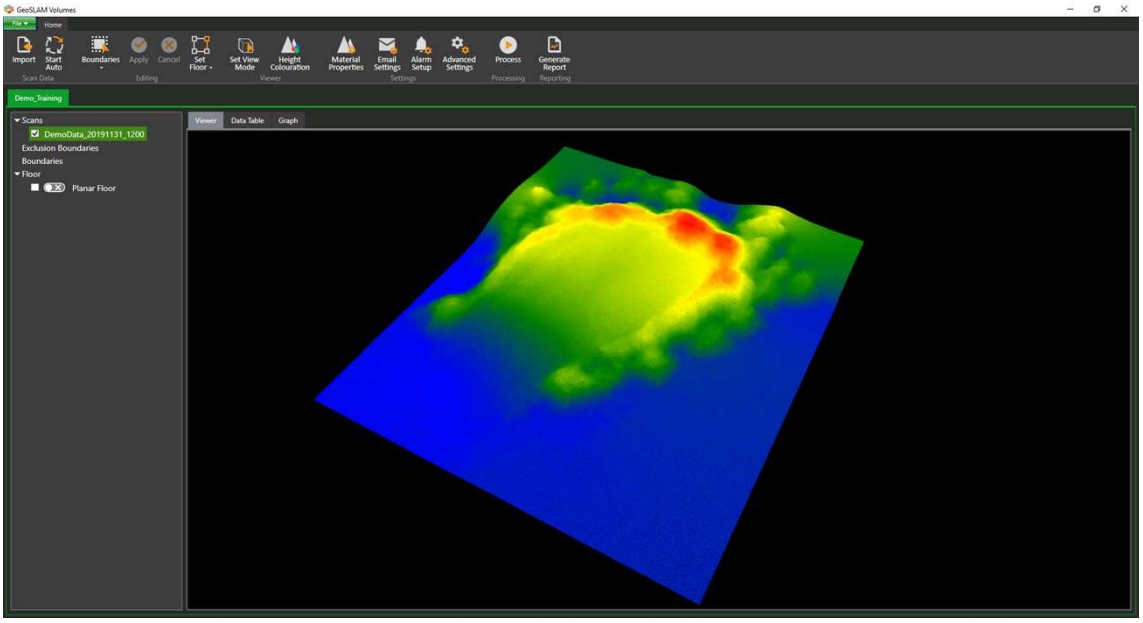


Figure 57. Imported data in the Project Tree and Viewer Window.

Start Auto.

To enable the automatic import and processing of data:

1. *Select* -> **Scan Data** tab-> **Start Auto** icon.
2. This will open the *Set Data Folder* dialog (Figure 58).

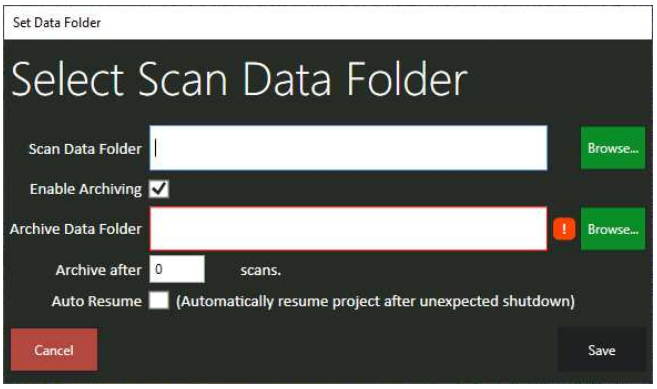


Figure 58. Automatic processing setup dialog.

Automatic processing setup functions are described in Table 8.

Function	Description
Scan Data Folder	*Browse* -> The user must select the data folder that will be monitored for new data.
Enable Archiving	Archiving will remove scan data from the project tree to improve system performance when a large number of scans (>50) are included in a project. Volumetric data are not removed from the data table or graph. It is only the pointcloud data that is archived.
Archive Data Folder	*Browse* -> The user *must select the* data folder where data will be archived.
Archive After	Enables the user to set the number of scans to display in the project. Scans in the project, greater than this value, will be moved to the archive folder and will no longer be visible in the Viewer window.
Auto Resume	Allows the user to set whether the current project will automatically start when the software is opened. This feature is beneficial for automated processing where the processing computer may be liable to intermittent loss of power.

Table 8. Automatic processing setup dialog options.

3. *Select* -> **Scan Data Folder** -> **Browse** -> to open the folder browser.
4. Browse to, and then select the scan data folder where the point cloud data will be stored.
5. *Select* -> **OK**, to continue.
6. If required, *Select* -> **Enable Archiving** -> *Checked.*
    a. *Select* -> **Archive Data Folder** -> **Browse** -> to open the folder browser.
    b. Browse to, and then select the scan data folder where the point cloud data will be archived. This will generally be a sub-folder of the processed data folder.
    c. *Select* -> **OK**, to continue.
    d. *Input* -> **Archive After** -> user defined value.

To maximise system performance, it is recommended to have **no more than 50 scans** loaded into a project at any given time.
Volumetric data displayed in the Data Table will **always** remain visible after archiving. Only the pointcloud data is removed from the viewer.

7. If required, *Check* -> **Auto Resume -> Checked**.
   This will ensure that the current project will be automatically started when opening the software where an unexpected shutdown has occurred.
8. *Select* -> **Save**, to continue or **Cancel** to return.
9. The software will now automatically, load, process and calculate volumetric data without any further user interaction required.

10. Automated data loading can be stopped at any time.
    a. *Select* -> **Scan Data** tab -> **Stop Auto** Icon.

Editing Tab

Boundaries

There are two types of boundaries that can be defined in the software.

1. **Exclusion boundary**: Exclusion boundaries are used to define an area outside of which all data is ignored. This can be useful when working in indoor stockpiles, or large outdoor stockpiles that are surrounded by infrastructure. Removing data that is not required will decrease processing times.
2. **Stockpile boundary**: Stockpile boundaries are used to define an area inside of which, data is assigned material properties and the volumetric and tonnage calculations are made.

Selecting the *Boundaries* icon will open a dropdown box (Figure 59) where the user can select either:

a. Create a Stockpile Boundary.
    i. Rectangular.
    ii. Polygonal.
    iii. Import a pre-existing boundary definition file.
b. Create an Exclusion Boundary.
    i. Rectangular.
    ii. Polygonal.
    iii. Import a pre-existing boundary definition file.

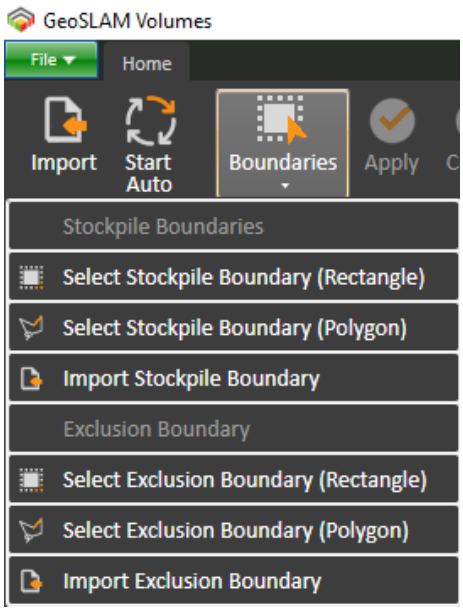


Figure 59. Boundary creation modes.

Exclusion Boundaries

Exclusion boundaries are used to define an area outside of which all data is ignored. This can be useful when working in indoor stockpiles, or large outdoor stockpiles that are surrounded by infrastructure. Removing data that is not required will decrease processing times.

Creating an Exclusion Boundary

To create a new exclusion boundary:

1. *Select* -> **Editing** tab -> **Boundaries** icon.
    a. *Select* -> **Select Exclusion Boundary (Rectangle)**, *or*
    b. *Select* -> **Select Exclusion Boundary (Polygon)**.

When the user selects either the Select Exclusion Boundary (Rectangle) or Select Exclusion Boundary (Polygon) icon the viewer window will automatically alter the view to a top-down 2D view for accurate definition of the boundary.

2. Highlight an area (either rectangular or a user define polygon) using the techniques described in Table 9. The area will be highlighted using a *Red* line.

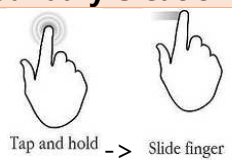
Function	Touchscreen	Mouse
Boundary Creation Mode		
Select (Rectangle)		Click and drag to create the rectangular area.
Select (Polygon)		*Click* -> to select a new vertex. *Double-Click* -> to close the polygon.

Table 9. Exclusion boundary creation options.

3. *Select* ->
    a. **Apply**, to create the newly defined exclusion boundary into the project, or
    b. **Cancel**, to cancel the creation.

Only 1 exclusion boundary can be open per project.

4. The user has the option to view the boundary using the checkbox next to the boundary name in the project tree.

5. The user has the option to make the boundary active by using the activation slider next to the boundary name in the project tree.

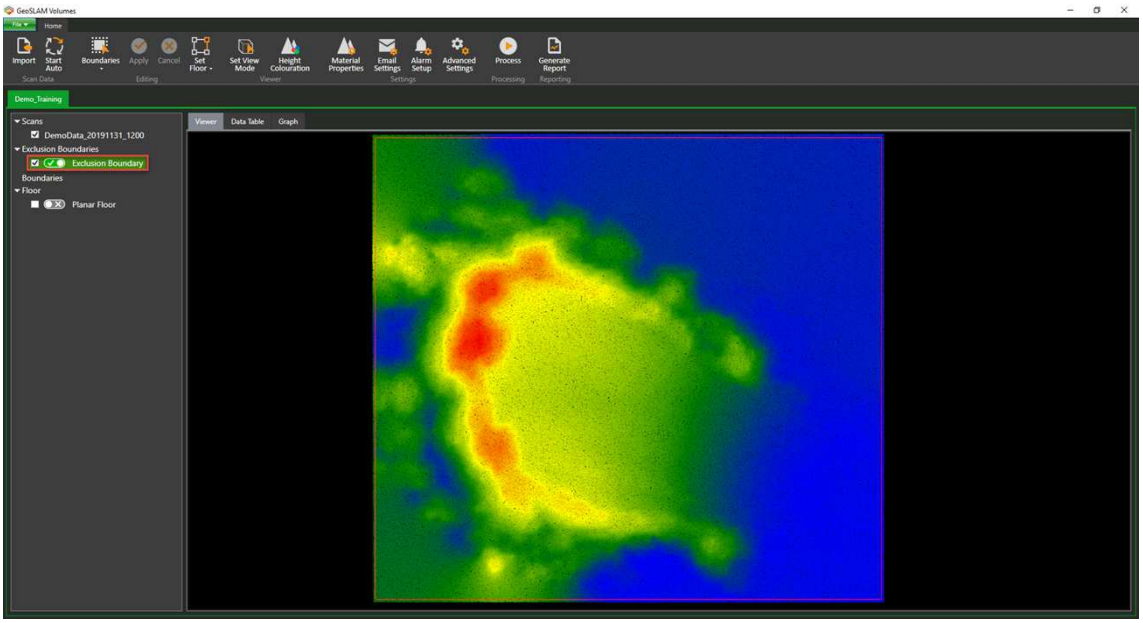


Figure 60. Exclusion boundary in the project tree.

Removing / Exporting an Exclusion Boundary

Once an exclusion boundary has been created the user has the option to either export or remove the boundary from the project via the project tree.

To remove or export a boundary:

1. Highlight the exclusion boundary in the project tree.
2. *Right-Click* -> boundary name. This will open a secondary options dialog (Figure 61).
3. *Select* ->
    a. **Export** -> to export and save the boundary to a DXF file, or
    b. **Remove** -> to delete it from the project.

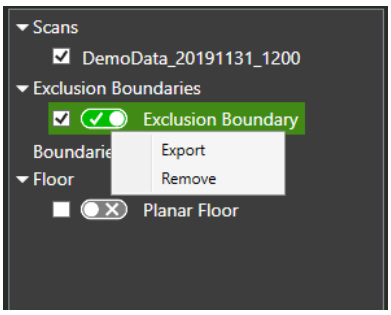


Figure 61. Exclusion boundary *Export / Remove* dialog.

4. Selecting to export the boundary opens the *Save Boundaries* dialog box.

a. *Browse* -> Chose a folder to store the boundary and enter a filename. The boundary is saved using the DXF file format.
b. *Select* -> **Save** to continue or **Cancel** to return.

5. Selecting to remove a boundary opens a confirmation dialog box.
   a. *Select* -> **Yes**, to delete the boundary.
   b. *Select* -> **No**, to cancel the deletion.

Loading an Exclusion Boundary

Exclusion boundary definition files in the AutoCAD DXF format can be loaded into the software. Currently, only 2D boundary definitions are utilised by the software. These files can be either created in *Volumes* using the method detailed above or using external software. The exclusion boundary file can contain multiple boundaries.

1. *Select* -> **Editing** tab -> **Boundaries** icon.
2. *Select* -> **Import Boundary**.
3. *Browse* to and *then Select* -> *Boundary file (DXF).*
4. Selecting a boundary file will open the *Boundaries Selection* dialog (Figure 62).

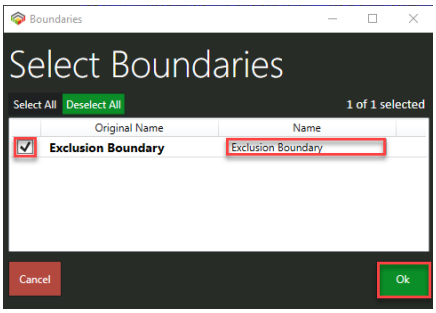


Figure 62. Import exclusion boundary selection dialog.

5. *Select* -> **Checkbox** -> *boundary name***.** Only 1 boundary can be chosen as an exclusion zone.
6. *Input* -> new *boundary name*, if required.
7. *Select* -> **OK**, to continue or **Cancel** to return.
8. *Select* -> **OK**, to continue at the confirmation dialog.

Stockpile Boundaries

Stockpile boundaries are used to define an area inside of which, data is assigned material properties and the volumetric and tonnage calculations are made.

Creating Stockpile Boundaries

To create a new stockpile boundary:

1. *Select* -> **Editing** tab -> **Boundaries** icon.

a. *Select* -> **Select Stockpile Boundary (Rectangle)**, *or*
b. *Select* -> **Select Stockpile Boundary (Polygon)**.

When the user selects either the Select Stockpile Boundary (Rectangle) or Select Stockpile Boundary (Polygon) icon the viewer window will automatically alter the view to a top-down 2D view for accurate definition of the boundary.

2. Highlight an area (either rectangular or a user define polygon) using the techniques described in Table 10. The area will be highlighted using a *Green* line.

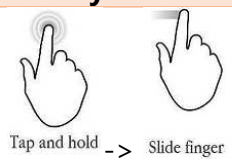
Function	Touchscreen	Mouse
Boundary Creation Mode		
Select (Rectangle)		Click and drag to create the rectangular area.
Select (Polygon)		*Click* -> to select a new vertex. *Double-Click* -> to close the polygon.

Table 10. Stockpile boundary creation options.

3. *Select* ->
   c. **Apply**, to create the newly defined boundary into the project, or
   d. **Cancel**, to cancel the creation.
4. Selecting *Apply* will open the *Create Boundary* dialog (Figure 63).

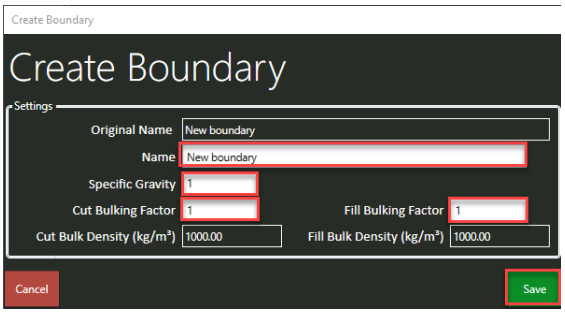


Figure 63. Create stockpile boundary dialog.

Create stockpile boundary options are described in Table 11.

Function	Description
Original Name	The name of the boundary created by the system.

Name	If required, the user can enter a new name for the boundary.
Specific Gravity	Specific gravity is a measure of density relative to the density of pure water. It is a dimensionless number.
Cut Bulking Factor / Fill Bulking Factor	The Bulking Factor accounts for the change in volume between in situ material and excavated material. It is entered as a ratio.
Cut / Fill Bulk Density	Bulk Density is defined as the mass of a bulk material divided by the volume occupied by that material. The bulk density of the material is automatically calculated by the software.

Table 11. Create boundary functions options dialog.

5. From the *Create Boundary* dialog:
    e. *Input* -> **Name** -> user defined name for the boundary.
    f. *Input* -> **Specific Gravity** of the material within the bounds of the area.
    g. *Input* -> **Cut Bulking Factor** of the material within the bounds of the area.
    h. *Input* -> **Fill Bulking Factor** of the material within the bounds of the area.
6. *Select* -> **Save**, to create the boundary or **Cancel**, to return.

Inputting values for the Specific Gravity, Cut and Fill Bulking Factor, the software will calculate the effective bulk density of the material that will be used in the tonnage calculation.

The new boundary will be visible in the *Boundaries* category in the project tree (Figure 64).

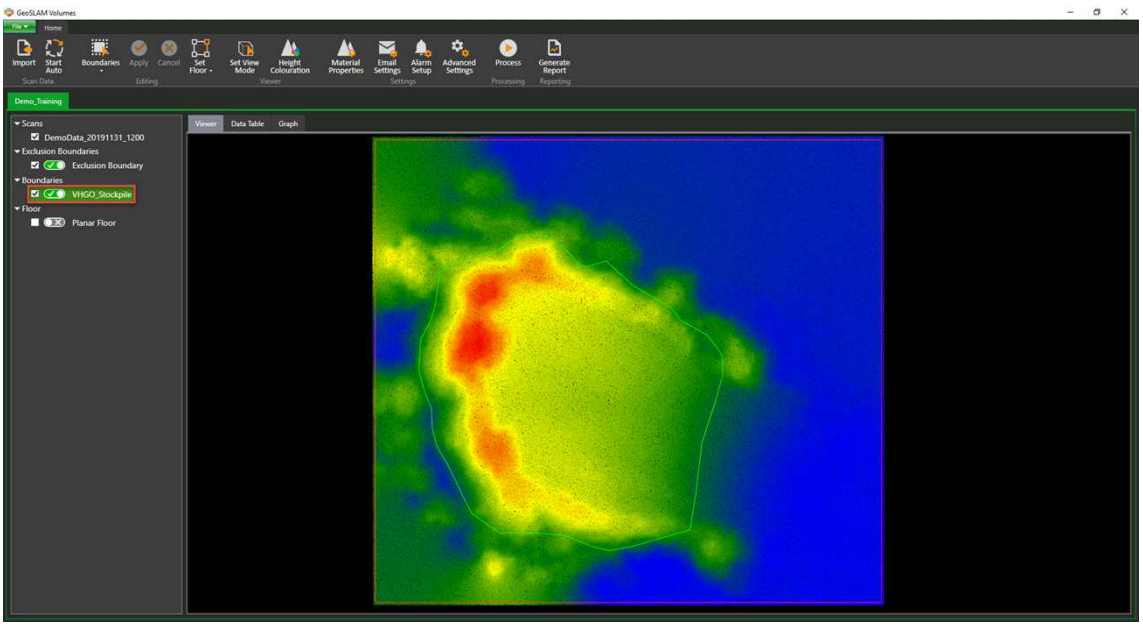


Figure 64. New boundary in the Project Tree.

Removing / Exporting Boundaries

Once a boundary has been created the user has the option to either export or remove the boundary from the project via the project tree.

To remove or export a boundary:

1. Highlight the boundary in the project tree.
2. *Right-Click* -> boundary name. This will open a secondary options dialog (Figure 65).
3. *Select* ->
    a. **Export** -> to export and save the boundary to a DXF file, or
    b. **Remove** -> to delete it from the project.

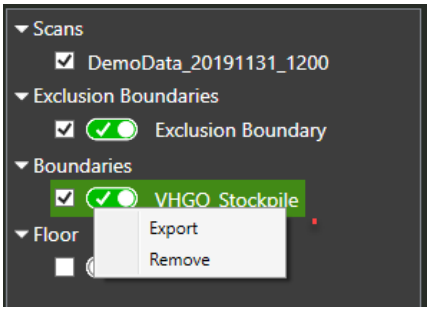


Figure 65. Boundary Export / Remove dialog.

4. Selecting to export the boundary opens the *Save Boundaries* dialog box.
5. *Browse* -> Chose a folder to store the boundary and enter a filename. The boundary is saved using the DXF file format.
6. *Select* -> **Save**, to continue or **Cancel**, to return.

Multiple boundaries can be exported at the same time by using Shift + Click and selecting the boundaries to export.

7. Selecting to remove a boundary open a confirmation dialog box.
    a. *Select* -> **Yes**, to delete the boundary.
    b. *Select* -> **No**, to cancel the deletion.

Loading Boundaries

Boundary definition files in the AutoCAD DXF format can be loaded into the software. Currently, only 2D boundary definitions are utilised by the software. These files can be either created in *GeoSLAM Volumes* using the method detailed above or using external software. The boundary file can contain multiple boundaries.

1. *Select* -> **Editing** tab -> **Boundaries** icon.
2. *Select* -> **Import Stockpile Boundary**.
3. *Browse* to and *then Select* -> *Boundary file (DXF).*

4. If a single boundary is contained in the file, the file will automatically load.
5. Where multiple boundaries are stores in a single DXF the user can select any or all boundaries via the *Boundaries Selection* dialog (Figure 66).

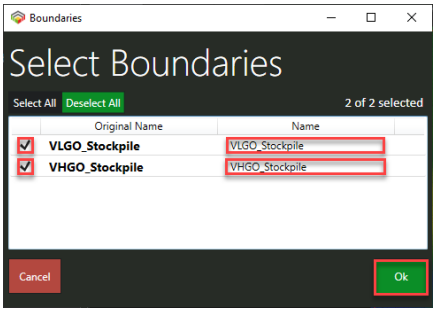


Figure 66. Import boundary selection dialog.

6. *Select* -> **Checkbox** -> *boundary name or Select* -> **Select All**.
7. *Input* -> new *boundary name*, if required.
8. *Select* -> **OK**, to continue or **Cancel** to return.
9. *Select* -> **OK**, to continue at the confirmation dialog.

When loading boundary files, the user must *Select* -> **Editing** tab -> **Material Properties** to define the required material properties.

Set Floor

The *Set Floor* function is used to define the reference plane from which all net volumetric analyses are calculated. 2 types are floor can be implemented.

1. Flat planar floor which can be either:
    a. Horizontal at a fixed height, or
    b. An inclined plane defined by selecting 3 points in the dataset.
2. Pre-existing reference data set.

Selecting the **Set Floor** icon will open a dropdown menu (Figure 67) where the user can select either:

a. Create a planar floor.
b. Import a file to use as the reference floor.

Figure 67. Reference floor creation tools.

Create Fixed Height Floor

When creating a fixed height planar floor, the software will, by default, automatically set the floor height at the lowest point in the data. To create a fixed height floor:

1. *Select* -> **Editing** tab -> **Set Floor** icon.
2. *Select* -> **Set Planar Floor**, to open the *Set Floor* dialog (Figure 68).

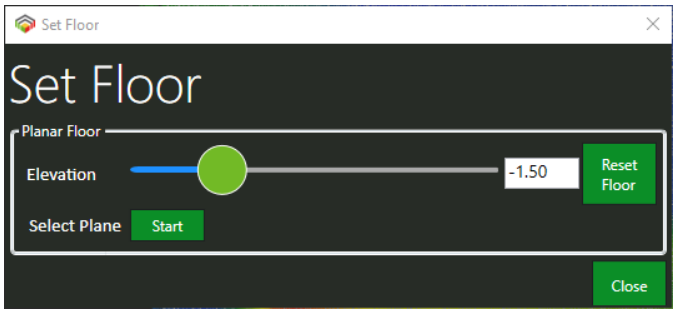


Figure 68. Set Floor height dialog.

3. In the Set Floor dialog, the user can choose to:
    a. Allow the software to define a horizontal plane at a value lower than the lowest point in the data (Figure 69).
    b. *Select* – **Reset** -> to define the horizontal plane at the lowest value in the point cloud (Figure 70).
    c. *Select* -> **Slider** or *Input* -> user defined value to define the horizontal plane at an arbitrary level (Figure 71).
4. *Select* -> **Close**, to continue.

The new floor will be visible in the *Floor* category in the project tree (Figure 69, Figure 70 and Figure 71).

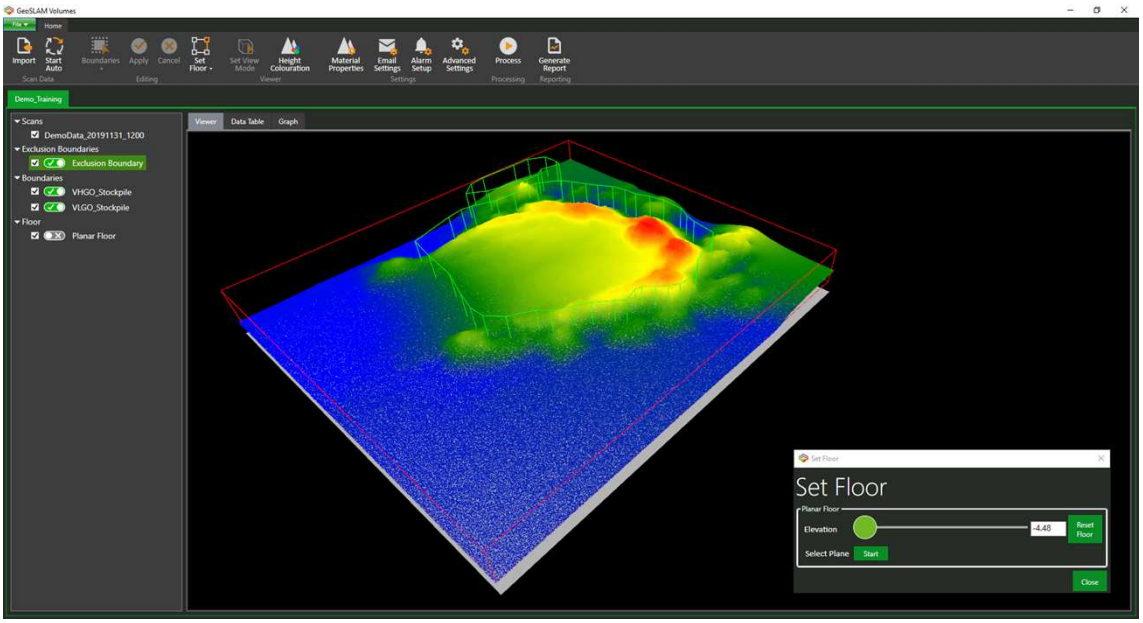


Figure 69. Floor level manually defined to a level below the lowest point in the data.

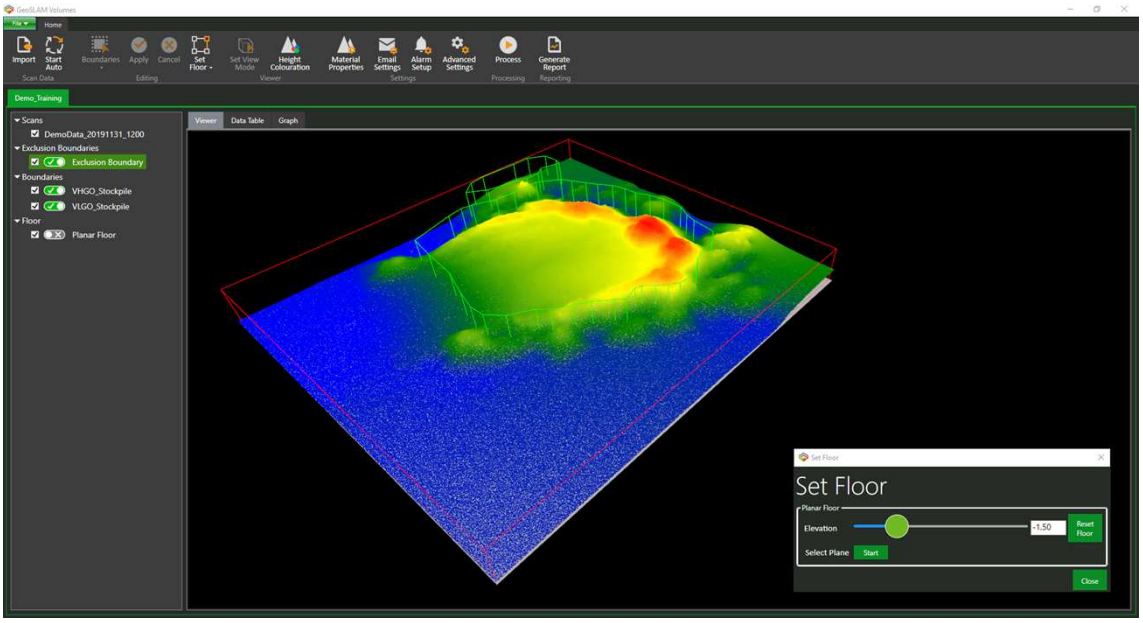


Figure 70. Automatically defined floor height set at the lowest data point.

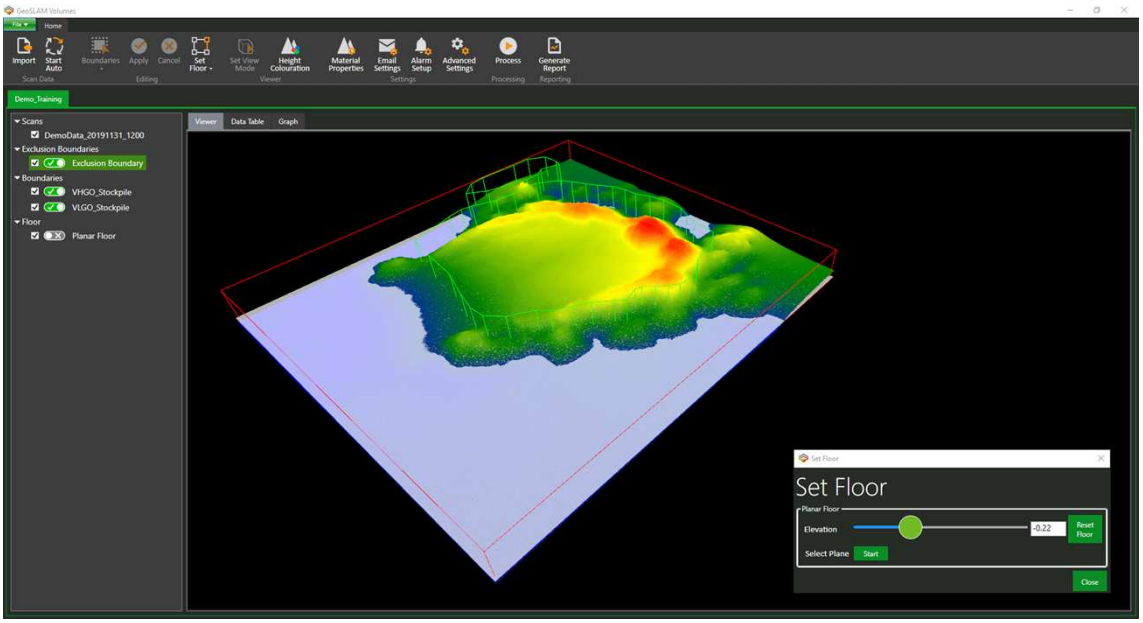


Figure 71. Defining the floor at a user defined level using the slider bar / input box.

Create an Inclined Floor

When creating an inclined floor, at least 1 dataset must be loaded and selected in the project.

To create an inclined floor:

1. *Select* -> **Editing** tab -> **Set Floor** icon.
2. *Select* -> Set **Planar Floor**, to open the *Set Floor* dialog (Figure 68).
3. *Select* -> **Planar Floor** -> **Select Plane** -> **Start**.
4. *Shift* + *Click* -> 3 points in the dataset (Figure 72).
5. Each selected point will be identified by a marker (Figure 73).
6. With 3 points selected the inclined plane will be displayed (Figure 74).

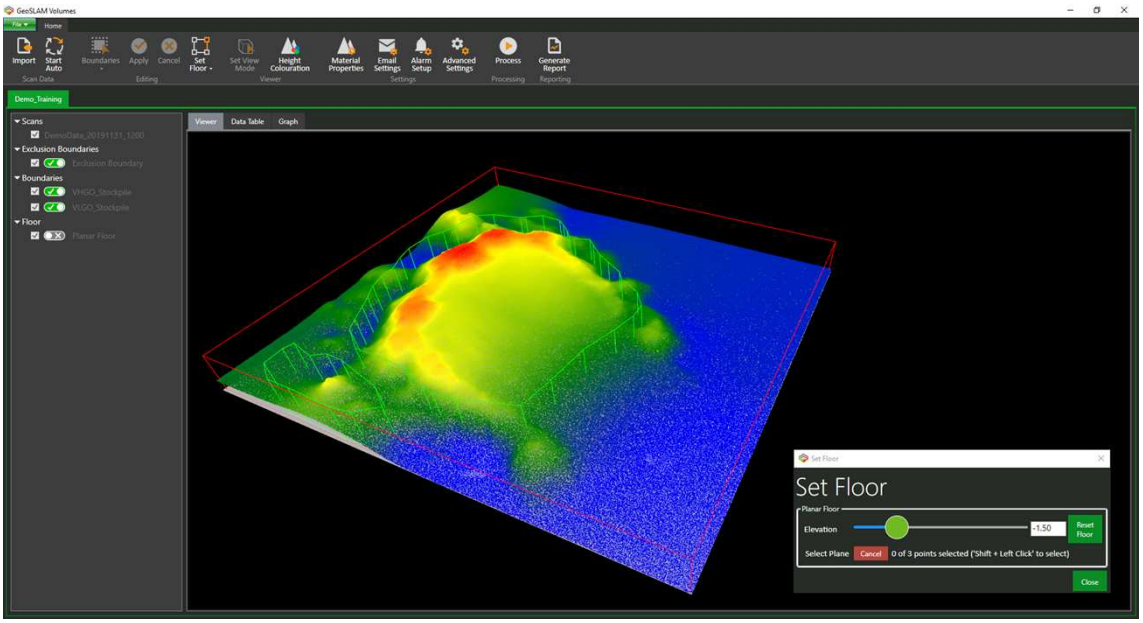


Figure 72. Creating an inclined plane by selecting 3 points from the dataset.

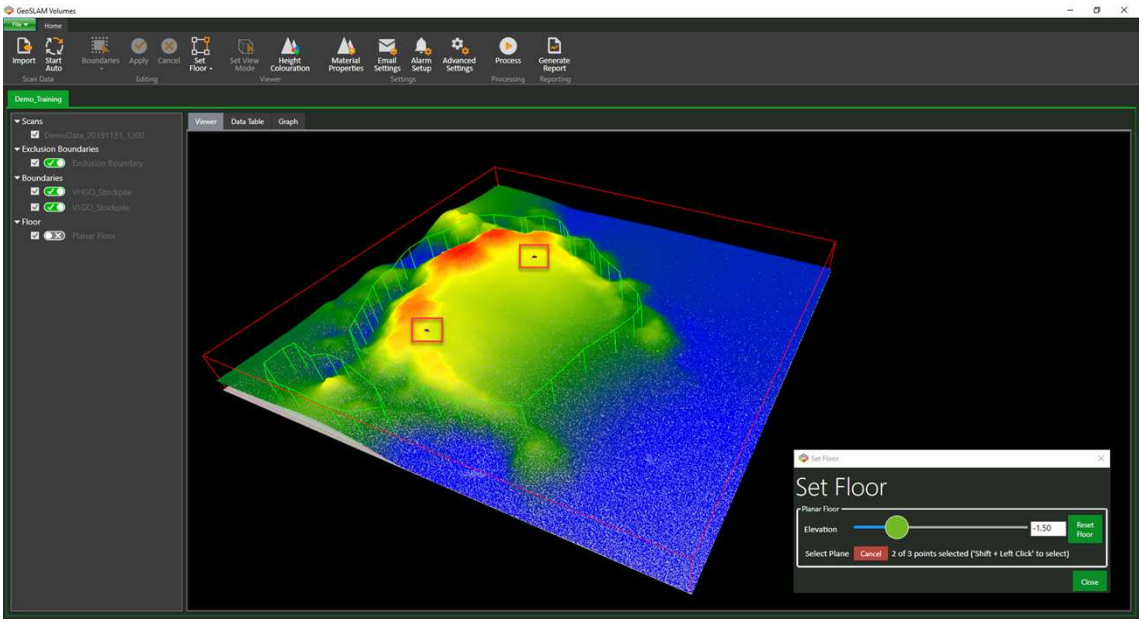


Figure 73. Selected points are highlighted in the dataset.

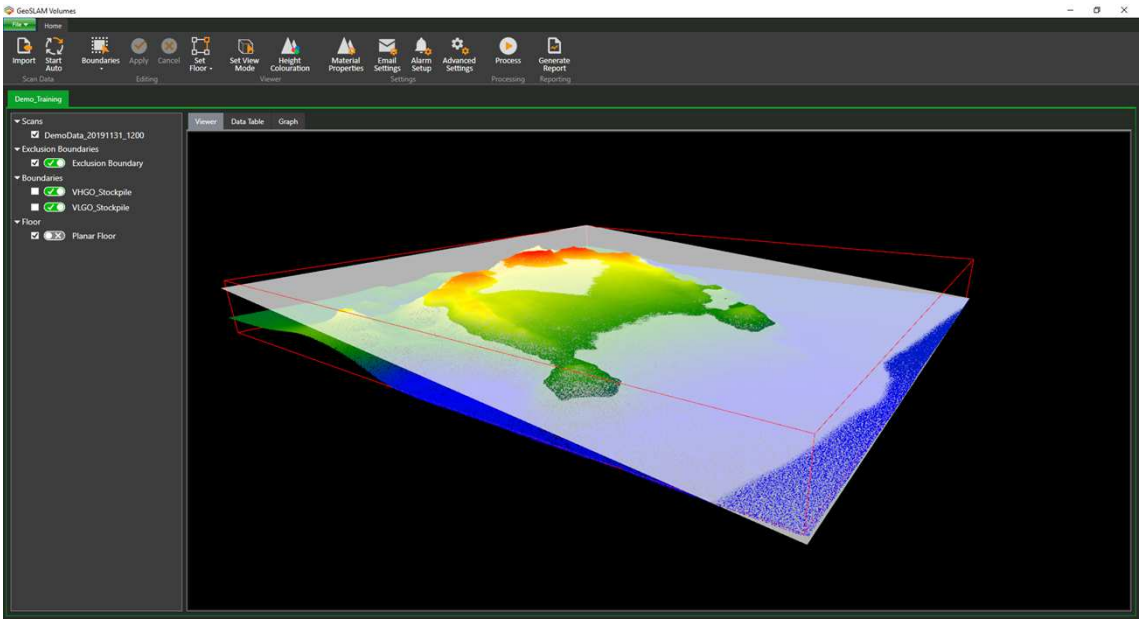


Figure 74. Created incline plane is displayed in the viewer.

At any time, the user can select the *Reset* icon in the *Set Floor* dialog to return the planar floor to a horizontal surface.

Import Floor

To import a reference from a pre-existing scan file:

1. *Select* -> **Editing** tab -> **Set Floor** icon.
2. *Select* -> **Import Floor**, to open the **Import Floor Scan** dialog (Figure 75).
    a. *Input* -> *Floor Name* (optional).
    b. *Select* -> -> **Browse** -> to open the folder browser.
    c. Browse to, and then select the scan data folder where the point cloud data will be stored.
    d. *Select* -> **Import**, to continue or **Cancel**, to return.

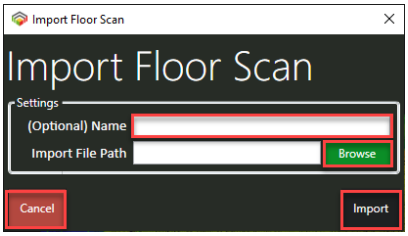


Figure 75. Import Floor Scan dialog.

3. The floor will be loaded and displayed in the Project Tree.

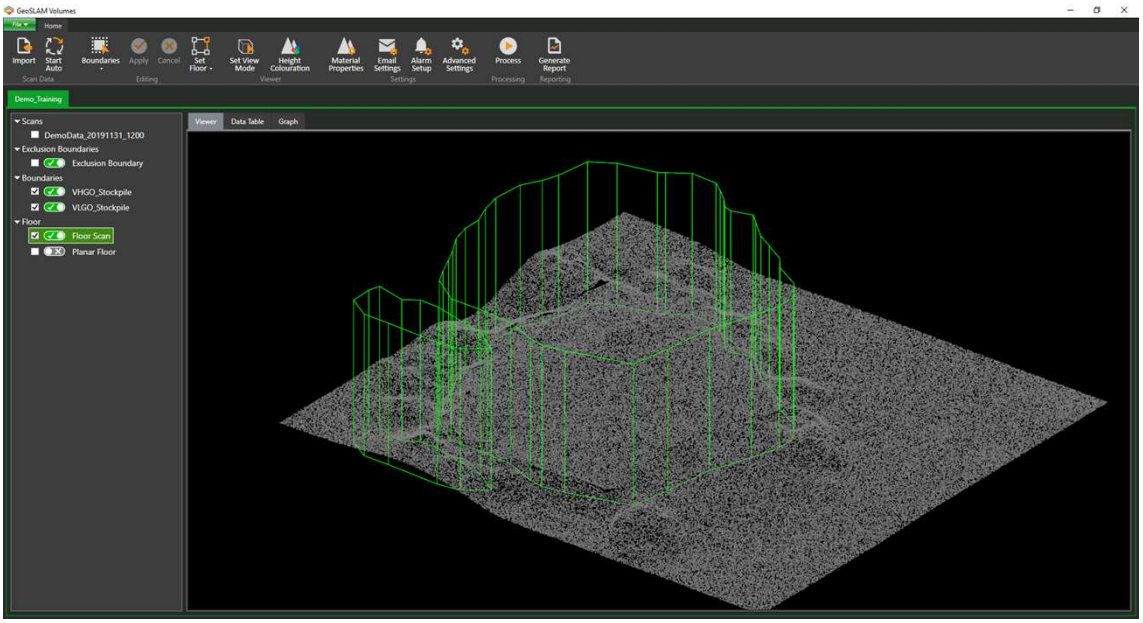


Figure 76. Floor scan loaded into the viewer and selected in the project tree.

Multiple floors can be displayed in the view. However, only a single reference floor can be selected as **Active** per project.

Viewer Tab

Toggle Projection

The user has the option to view the data in the *Viewer* windows using either:

a) 3D Perspective projection, or
b) 3D Orthometric projection.

To change the view perspective:

1. *Select* -> **Viewer** tab -> **Toggle Projection** icon to switch between the 2 display modes.

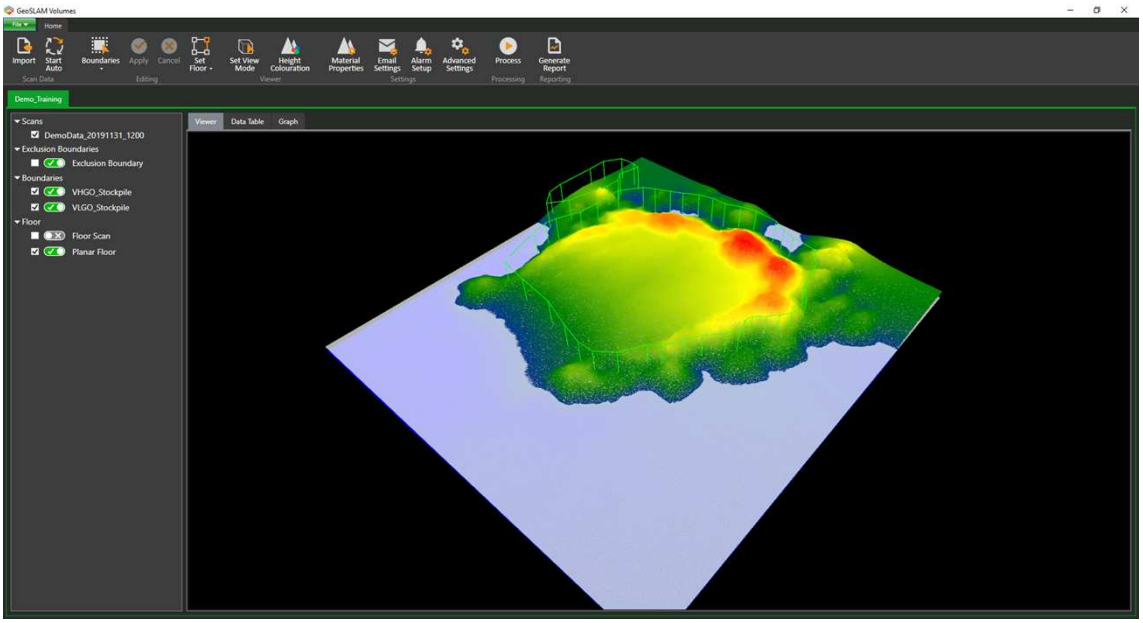


Figure 77. Example of Perspective view.

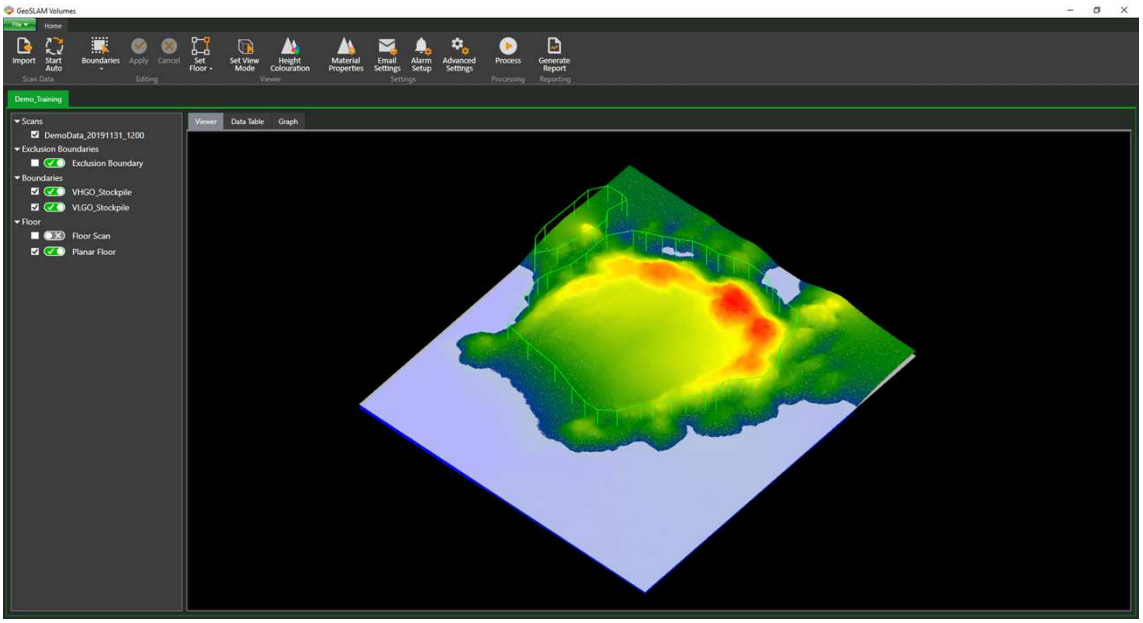


Figure 78. Example of Orthometric view.

In a Perspective view, objects that are further away look smaller, even if they are the same size. In an Orthographic view, objects that are the same size look the same size, even if they are further away. Humans see the world around us in Perspective, but Orthographic is usually easier to think about when positioning or modelling objects.

Height Colouration

The Height Colouration feature allows for the selection of the colourmap used to colour any loaded pointcloud elevation data. The following default shaders are available:

- **Single Colour Shader**. All points are displayed as Grey.
- **Colour Ramp Shader** displays ramps colour from blue for low elevations to red for the highest elevations by default.

To alter the colour view:

1. *Select* -> **Viewer** tab -> **Height Colouration** icon.
2. This will open the *Set Colour* dialog (Figure 79).

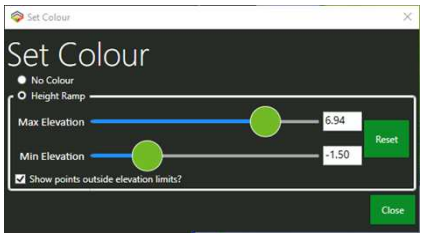


Figure 79. Set Colour dialog.

3. *Select* -> **No Colour**, to view the data using a single colour (Figure 80).
4. *Select* -> **Height Ramp**, to enable the colour ramp shader.
5. The user can then manually set the **Max Elevation** (coloured Red) and **Min Elevation** (coloured Blue) by either using:
    a. *Select* -> **Slider** or *Input* -> *user defined value* to manually set the values, or
    b. *Select* -> **Reset**, to automatically set the max and min values.

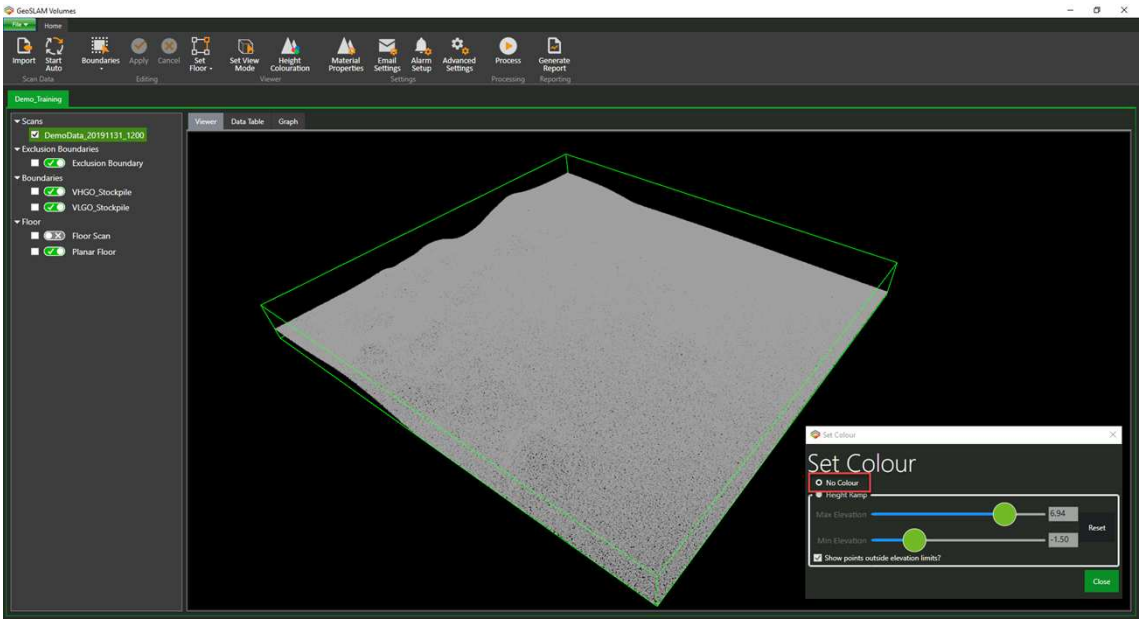


Figure 80. Data displayed as a single colour with no shader applied.

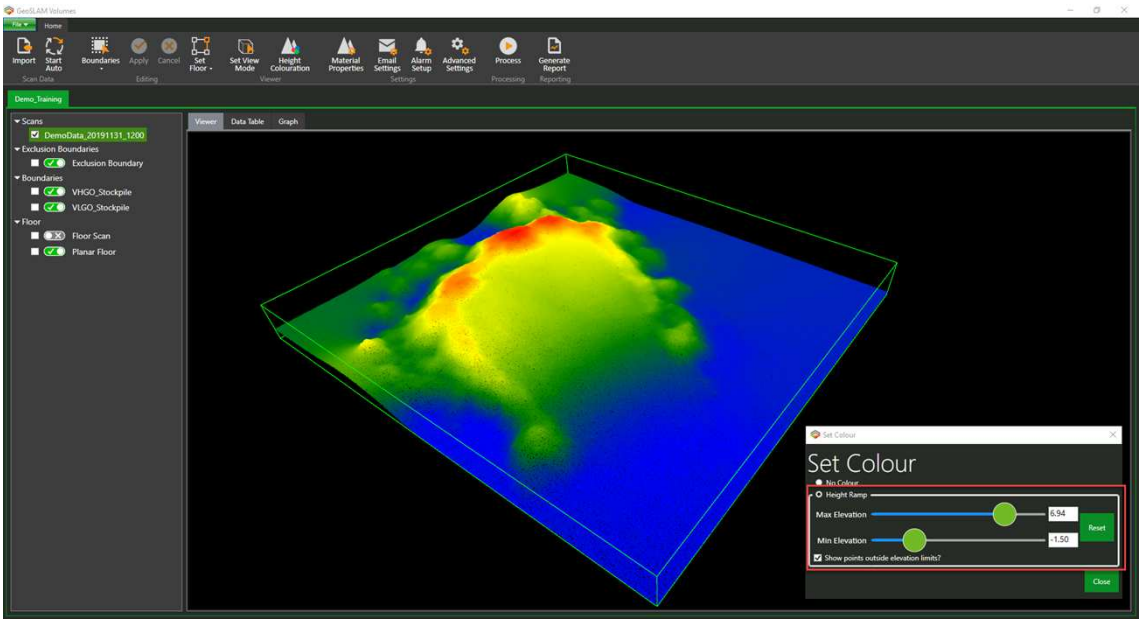


Figure 81. Data coloured using the Colour Ramp Shader.

6. *Check / Uncheck* -> **Show points outside elevation limits,** to either:
    a. display any points within the loaded dataset that are outside of the current shader values (these points will be displayed in Grey), or
    b. Hide any points within the loaded dataset that are outside of the current shader values.
7. *Select* -> **Close**, to continue.

The Set Colour function only affects the colourisation of the points within the data viewer window. It does not affect the data used in the volumetric calculations.

Settings Tab

Material Properties

The set materials properties dialog allows the user to:

1. Set / alter the Specific Gravity and Cut and Fill Bulking Factors for all materials in the project, or
2. Set / alter the Specific Gravity and Cut and Fill Bulking Factor for materials within a specific boundary.

Choosing to set a *Project Level* Specific Gravity and Bulking Factors, should be used when no boundary definitions are created. If there are any active boundaries in the project, then the data table will show Volumes and Tonnage for each boundary in the project. To set the material properties:

1. *Select* -> **Settings** tab -> **Set Material Properties** icon.
2. This will open the Set Material Properties dialog (Figure 82).

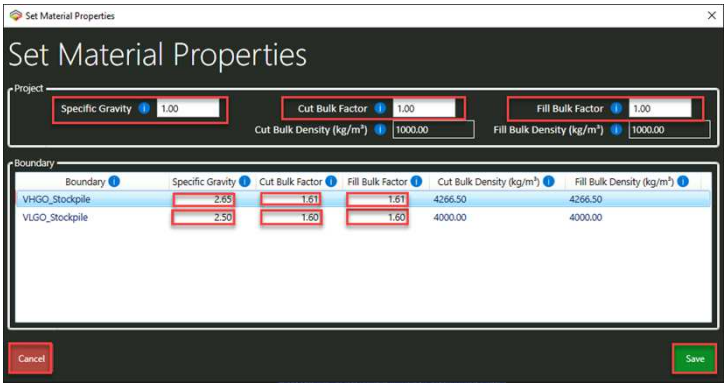


Figure 82. *Set Material Properties* dialog.

3. To set / change the global project properties:
    a. *Input* -> **Specific Gravity**.
    b. *Input* -> **Cut Bulk Factor**.
    c. *Input* -> **Fill Bulk Factor.**
    d. *Select* -> **Save**, to continue or **Cancel**, to return.
4. To set / change the individual boundary properties:
    a. *Select* -> *Boundary*.
    b. *Input* -> **Specific Gravity**.
    c. *Input* -> **Cut Bulk Factor**.
    d. *Input* -> **Fill Bulk Factor**.
    e. *Select* -> **Save**, to continue or **Cancel**, to return.

Specific Gravity and Bulking Factors for a range of natural materials is provided in Table 12.

Material	Specific Gravity	Bulking Factor
Clay (Low PI)	1.65	1.30
Clay (High PI)	2.10	1.40
Clay and Gravel	1.80	1.35
Sand	2.00	1.05
Sand & Gravel	1.95	1.15
Gravel	2.10	1.05
Chalk	1.85	1.50
Shales	2.35	1.50
Limestone	2.60	1.63
Sandstone (Porous)	2.50	1.60
Sandstone (cemented)	2.65	1.61
Basalt	2.95	1.64
Granite	2.41	1.72

Table 12. Examples of Specific Gravity and Bulking Factor for various materials.

The table provides example values for the specific gravity and bulking factor. It is important that site specific values are entered by the user to obtain reliable tonnage values.

Email Settings

GeoSLAM Volumes provides the user with the ability to set up email alerts when either min and /or max, net volume, net tonnage or a height above the reference floor values are met. The *Project Settings* dialog (Figure 83) prompts the user to enter the host and server details from where the email alerts are sent.

To configure the email alert settings:

3. *Select* -> **Settings** tab -> **Email Settings** icon.
4. This will open the *Project Settings* dialog (Figure 83).
5. *Input* -> **Username** -> *username for the email account.*
6. *Input* -> **Password** -> *password for the email account.*
7. *Input* -> **Host** -> *user specified email server*, e.g., smtp.office365.com.
8. *Input* -> **Port**-> *user specified port number.* Port 587 is the standard port assignment for SMTP email.
9. *Input* -> **Sending Email Address** -> The email address from which alerts are sent.
10. *Enable* -> Use SSL. SSL (Secure Sockets Layer) is the standard security technology for establishing an encrypted link between a server and a client.

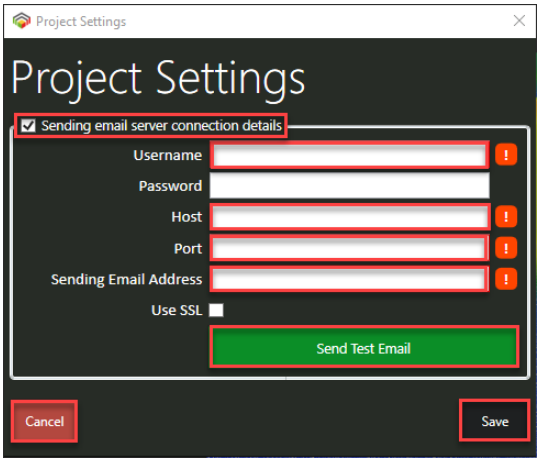


Figure 83. *Project Settings* dialog to setup system email alerts.

Testing the email settings

To test the settings that have been entered:

1. Ensure that the correct settings have been entered into the required boxes.
2. *Select* -> **Send Test Email**.
3. *Select* -> **Email Test** -> **Yes**, at the confirmation dialog.
4. If the test has been successful, a confirmation dialog will be displayed, and a test email will be sent to the *Sending Email Address* mail account.
5. *Select* -> **OK**, to continue.
6. Open a mail browser and confirm that the email has been received (Figure 84).

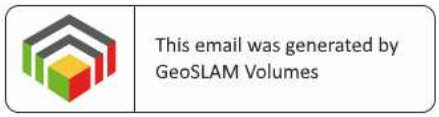


Figure 84. Email response from a successful test email.

7. If the test has been unsuccessful, a confirmation dialog box will be displayed stating that the test email was not able to be sent. Re-check the settings.

Alarm Setup

GeoSLAM Volumes provides the user with the ability to set up email alerts when either min and / or max, net volume, net tonnage or a height above the reference floor values are met. The *Alarm Setup* dialog (Figure 85) allows the user to set both the recipients of the alert emails together with the bounding values that will trigger the alert email. Trigger values are set per boundary.

To configure the alarm levels:

1. *Select* -> **Settings** tab -> **Alarm Setup** icon.

2. This will open the *Alarm Setup* dialog (Figure 85).
3. *Input* -> **Receiving Email Addresses** -> email addresses of the people to receive the email alerts.

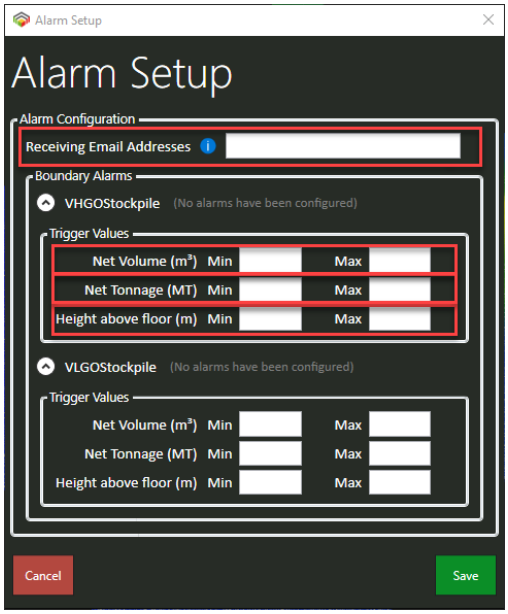


Figure 85. *Alarm Setup* dialog.

There is no limit on the number of people who will receive the alerts. Individual addresses must be separated by a comma (',').

4. Input -> **Trigger Values**:
   a. **Net Volumes** -> *Min* and / or *Max*, as required.
   b. **Net Tonnage** -> *Min* and / or *Max*, as required.
   c. **Height Above the Floor** -> *Min* and / or *Max*, as required.
5. *Select* -> **Save**, to continue.

The units for the Net Volume, Net Tonnage and Height above floor are set in the application settings.

Advanced Settings

Selecting the *Advanced Settings* icon will open the *Advanced Settings* dialog (Figure 86). This will allow users to define the project level settings for:

- Processing Settings.
- Date Settings format.

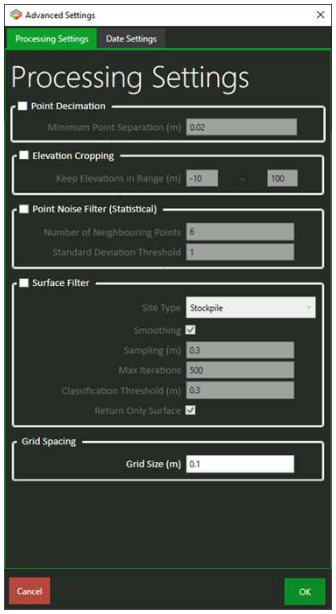


Figure 86. *Advanced Settings* dialog.

Processing Settings

Laser scans typically generate point cloud datasets of varying point densities. Additionally, measurement errors lead to sparse outliers which may corrupt the results. This complicates the estimation of local point cloud characteristics such as the Volume metrics, leading to incorrect volume estimates. Some of these irregularities can be solved by performing a statistical analysis on each point and trimming those which do not meet a certain criterion.

The *Processing Settings tab* (Figure 87) allows the user to define functions, and their settings, to overcome issues with the point cloud. Processing functions are:

1. Point Decimation.
2. Elevation Cropping.
3. Point Noise Filter (Statistical).
4. Surface Filter.

Any or all of these processing functions can be utilised for a given project.

Processing will run in the order in the settings dialog. For example:

1. If Point Decimation and Surface Filtering are checked, then the point cloud will be decimated and then a surface filter applied to the decimated data.
2. If Elevation Cropping and the Point Noise Filter are checked, then the data will have the elevation crop applied and then the point noise filter applied to the cropped data.

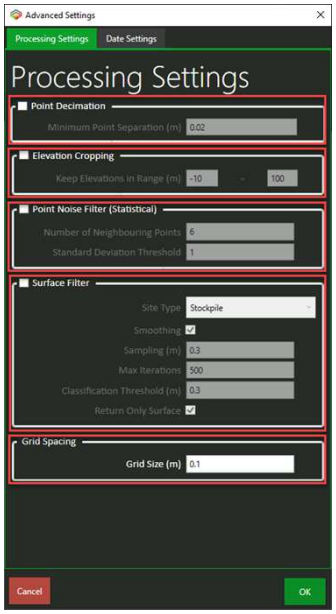


Figure 87. Advanced Processing Settings.

Additionally, the *Advanced Settings* dialog allows the user the ability to set the grid spacing that will be used to create the raster from which volumetric calculations are made.

Processing of data is only undertaken during data import.

Point Decimation

Decimation is the process of discarding points from the data to improve performance and reduce disk usage. In GeoSLAM Volumes, points are decimated based on their geometric separation from another point. In this way, areas of high point density are thinned whereas areas of sparse point spacing are not affected.

To enable point decimation:

1. *Select* -> **Advanced Settings** icon, to open the *Advanced Settings* dialog (Figure 87).
2. *Select* -> **Processing Settings** tab.
3. *Select* -> **Processing Settings** -> **Point Decimation** -> **CHECKED**.
4. *Input* -> **Minimum Point Spacing** -> *user defined* value (**default=0.02m**).
5. *Select* -> **Save**, to continue.

The units of the minimum point separation is defined by *Global Unit*.

The effect of point decimation can be seen by observing the stockpile scan shown before decimation (Figure 88-Upper) and after decimation (Figure 88-Lower).

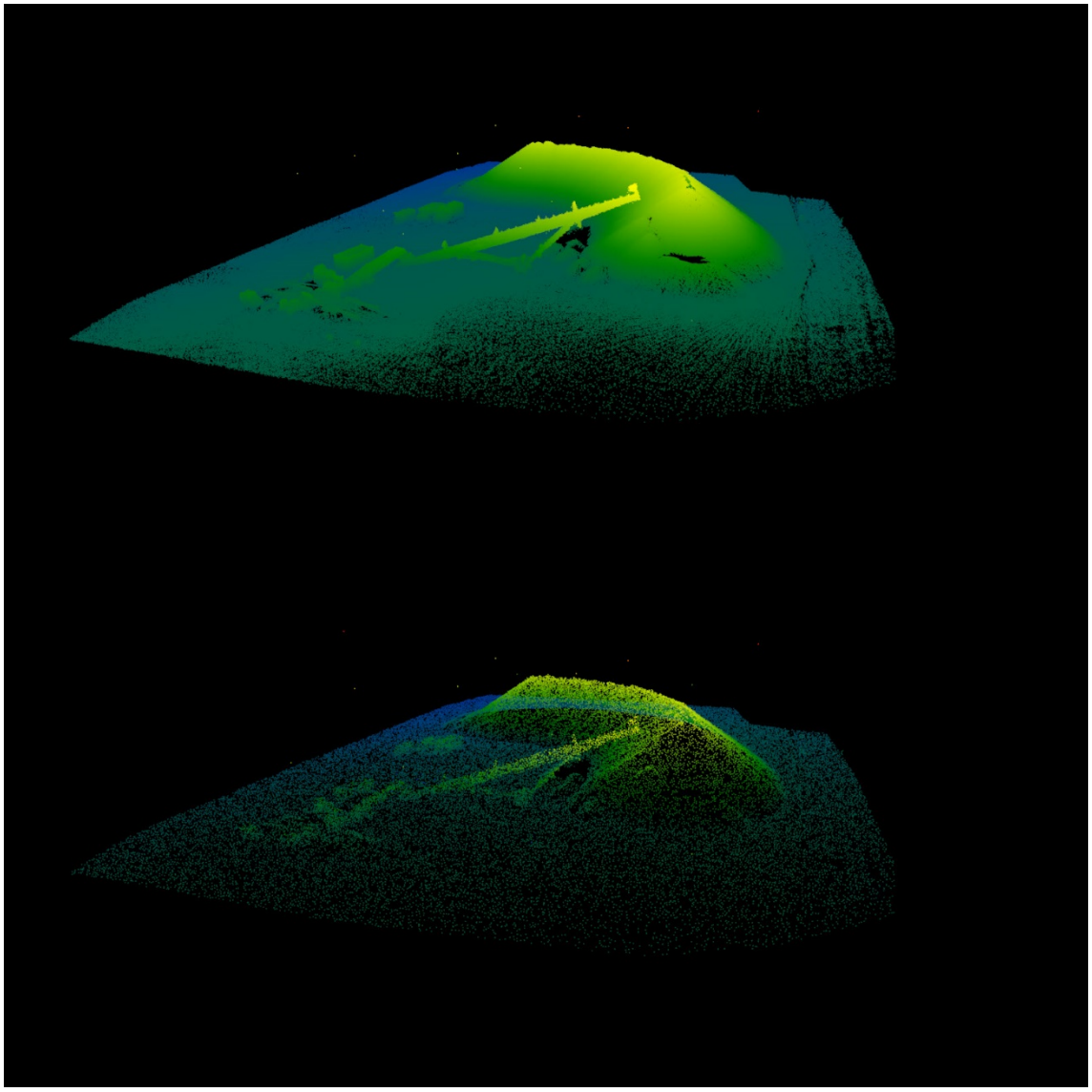


Figure 88. Raw Point Cloud before processing (upper) and after decimation (lower).

Elevation Cropping

Points that lie below the ground surface or points that appear well above the ground surface, e.g., reflections from windows, can be removed by settings the elevation cropping filter.

To enable elevation cropping:

1. *Select* -> **Advanced Settings** icon, to open the *Advanced Settings* dialog (Figure 87).
2. *Select* -> **Processing Settings** tab.
3. *Select* -> **Elevation Cropping** -> **CHECKED**.
4. *Input* -> **Keep Elevations in Range** -> *user defined value* (**default: -10 -> +100**).
5. *Select* -> **Save**, to continue.

The units of the minimum point separation is defined by *Global Unit.*

The effect of elevation cropping can be seen by observing the stockpile scan shown before decimation (Figure 89-Upper) and after elevation cropping (Figure 89-Lower).

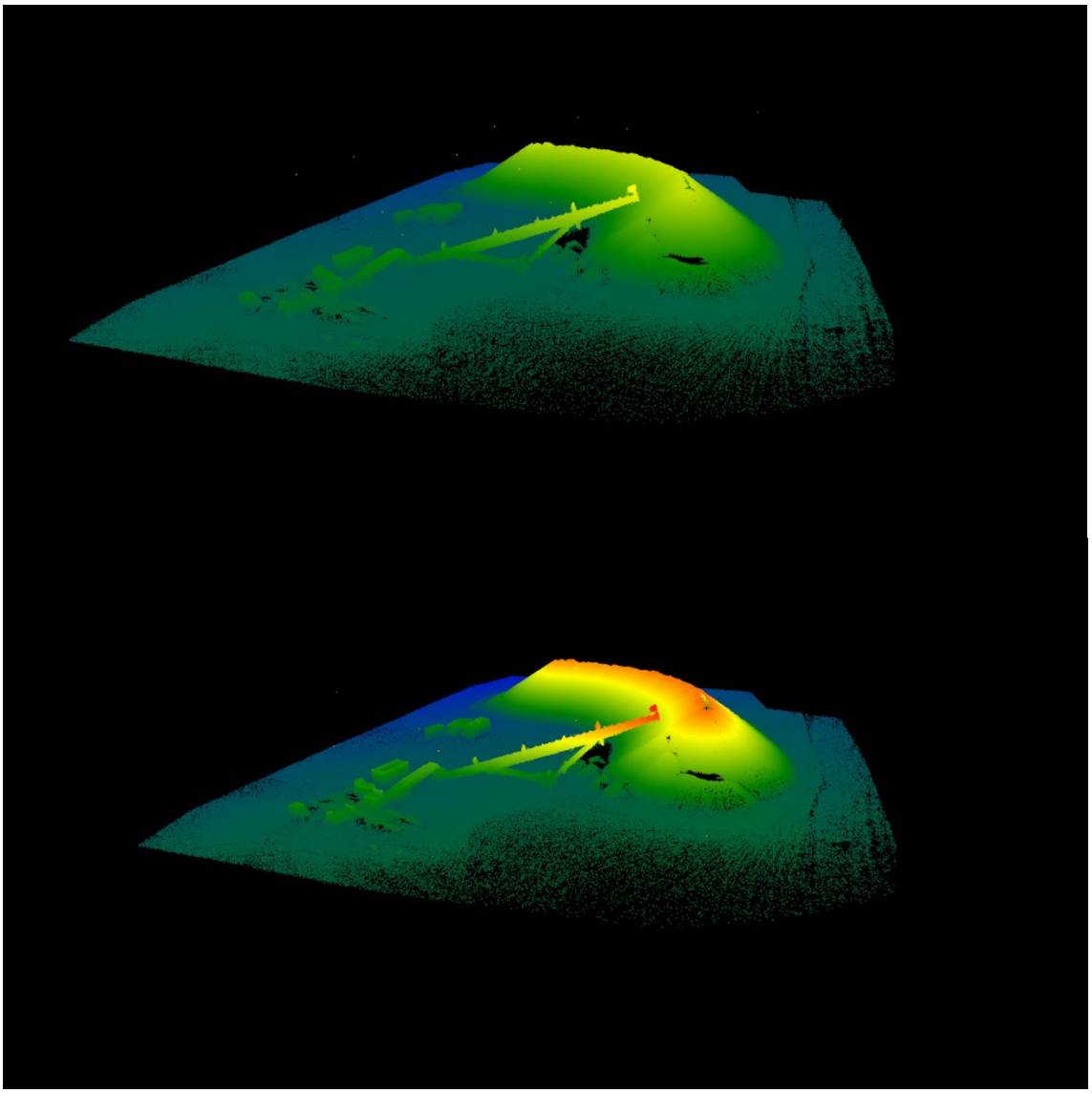


Figure 89. Raw Point Cloud before processing (upper) and after elevation cropping (lower).

<u>Point Noise Filter (Statistical)</u>

To minimise erroneous points *GeoSLAM Volumes* utilises a Statistical filtering algorithm. This algorithm is based on the computation of the distribution of point to neighbours' distances in the input data. For each point, the algorithm computes the distance to its neighbours. The number of neighbour points is defined by the *Number of Neighbouring Points* input. The mean and standard deviation of all point distances is then calculated. Any point whose standard deviation is greater than the Standard Deviation Threshold is assumed to be an outlier and removed from the data.

To enable point noise filtering:

1. *Select* -> **Advanced Settings** icon, to open the *Advanced Settings* dialog (Figure 87).
2. *Select* -> **Processing Settings** tab.
3. *Select* -> **Point Noise Filtering (Statistical)** -> **CHECKED.**
4. *Input* -> **Number of Neighbouring Points** -> *user defined* value (**default=6**).
5. *Input* -> **Standard Deviation Threshold** -> *user defined* value (**default=1**).
6. *Select* -> **Save**, to continue.

The effect of point noise filtering can be seen by observing the stockpile scan shown before decimation (Figure 90-Upper) and after noise filtering (Figure 90-Lower).

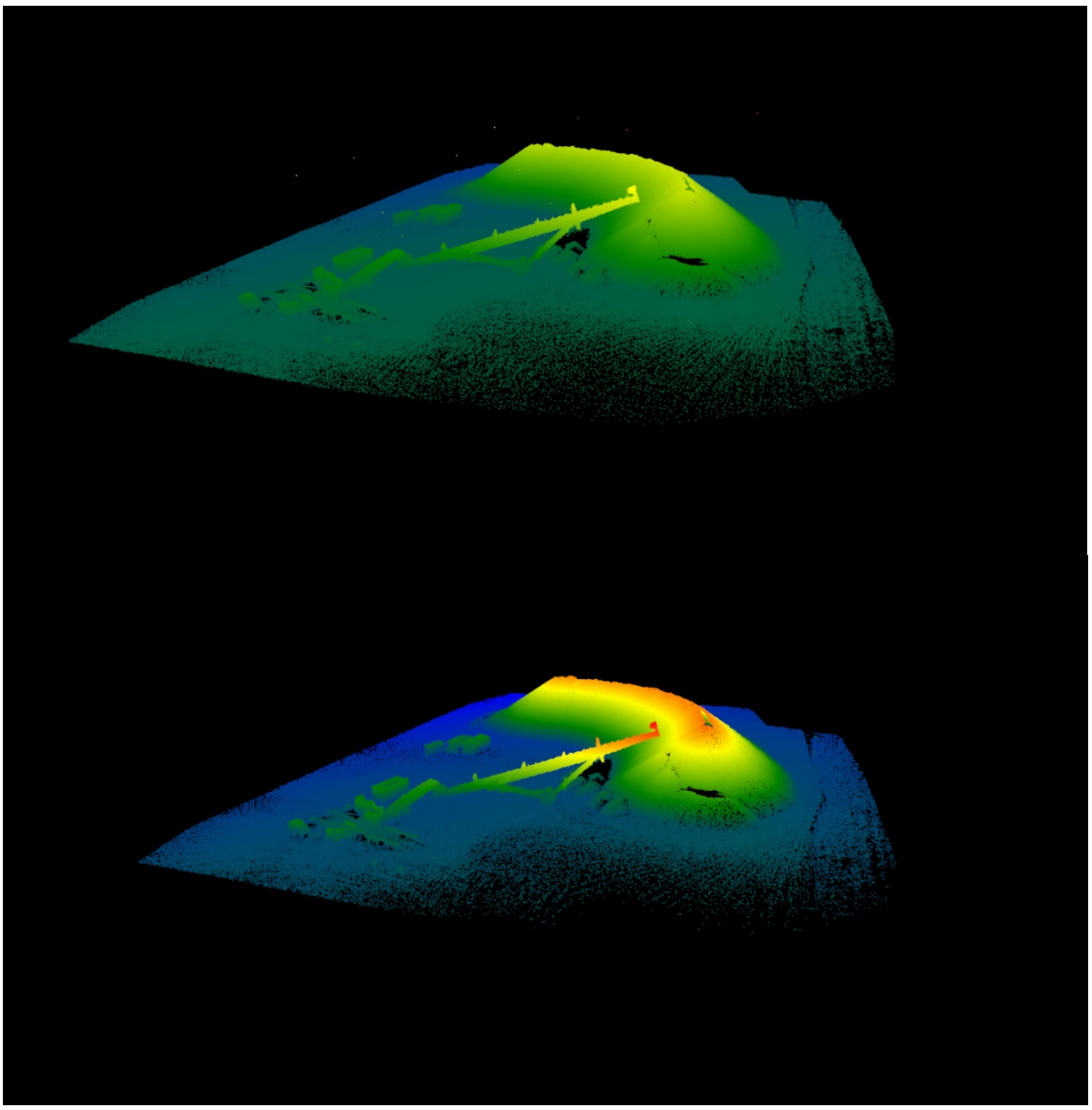


Figure 90. Raw Point Cloud before processing (upper) and after point noise filtering (lower).

<u>Surface Filter</u>

The surface filter attempts to extract only ground points from the point data. In this way, it created a Digital Terrain Model from the original pointcloud that represents the Digital Surface Model. This filter is useful for removing infrastructure, vehicles, and people.

The algorithm attempts to fit a surface to lowest points in the pointcloud. To allow the surface to follow the overall shape of the pointcloud the user must set the steepness of the terrain using the **Site Type** variable. This user can choose between areas of flat terrain typically found in earthwork locations or relatively steeper terrain such as that found in a typical stockpile environment; by selecting either *Stockpile* or *Earthworks* in the **Site Type** variable.

To help the surface fitting, the user must set the grid size covered by each cell of the surface, set in the *Sampling* variable. The algorithm will then iterate through the pointcloud to find the best result. The number of iterations is set in the *Max Iterations* variable.

Finally, having determined the best surface that matches the lowest surface in the pointcloud, the algorithm classifies points into Ground and Non-Ground. The user can choose to keep the non-ground points by de-selecting the Return-only-Surface.

To enable surface cropping:

1. *Select* -> **Advanced Settings** icon, to open the *Processing Settings* dialog (Figure 87).
2. *Select* -> **Processing Settings**.
3. *Select* -> **Surface Filter** -> **CHECKED**.
    a. *Select* -> **Site Type** -> **Earthworks** or **Stockpile**.
    b. *Select* -> **Smoothing** -> **CHECKED** or **UNCHECKED**, e.g., *CHECKED*.
    c. *Input* -> **Sampling** -> *user defined value*, e.g., 0.3.
    d. *Input* -> **Max Iterations** -> *user defined value*, e.g., 500.
    e. Input -> **Classification Threshold** -> *user defined value*, e.g., 0.3.
    f. Select -> **Return Only Surface** -> **CHECKED** or **UNCHECKED**, e.g., *CHECKED*.
4. *Select* -> **Save**, to continue.

The effect of the surface filtering can be seen by observing the stockpile scan shown before decimation (Figure 91 -Upper) and after noise filtering (Figure 91-Lower).

Grid Spacing

The grid spacing parameter allows the user to set the grid size used to create the raster. The following factors should be considered when specifying the cell size:

- The spatial resolution of the input data.
- The application and analysis that is to be performed.
- The size of the resultant database compared to disk capacity.
- The desired response time.

Typically, a grid size of 4 times the point spacing will achieve adequate results.

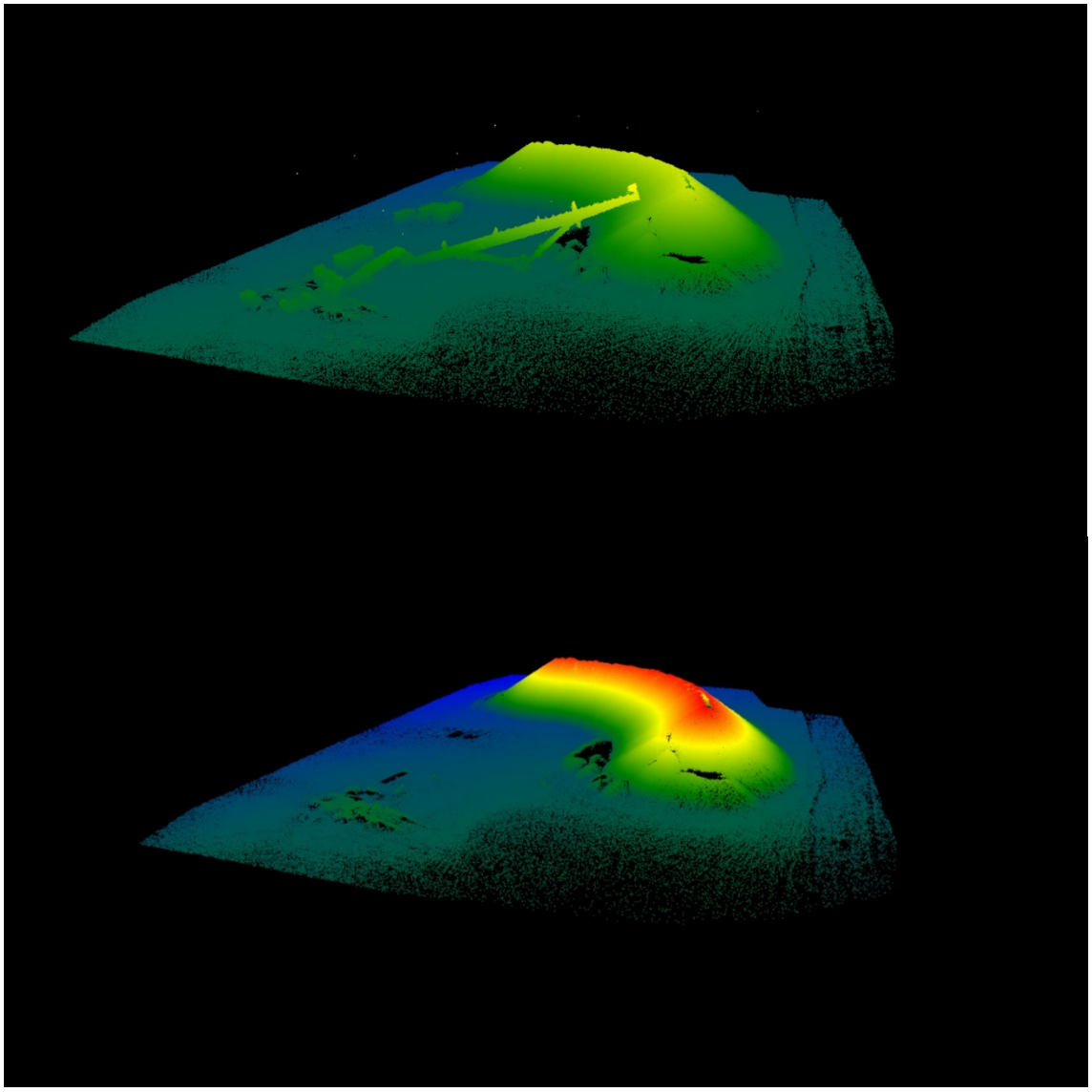


Figure 91. Raw Point Cloud before processing (upper) and after surface filtering (lower).

Date Settings

Typically, the *Project Data Settings* are configured and set in the program applications settings (more details are provided in the *Application Settings* section).

If a format has been defined and the *Extract Date from Scan Filename* configured in the application settings then all new projects will, by default, use these settings. The user has the ability to turn off the setting and / or define a new format in the *Date Setting* tab.

To set modify the default date setting for a specific project:

1. *Select* -> **Advanced Settings** icon, to open the *Processing Settings* dialog (Figure 87).
2. *Select* -> **Date Settings** tab (Figure 92).

Figure 92. Reporting Setup tab.

3. *Select* -> **Extract Date From Scan Filename** -> **CHECKED / UNCHECKED**.
4. *Input* -> **Format** -> *user defined format*.
    a. It is recommended to use filenames with the date string at the end of the filename, e.g., *stockpile1_16012021_1325.csv*
    b. In this case the user would enter -> **ddMMyyyy_HHmm*

Details of the Date Formats that can be used to define the template are given in Table 6.

5. *Select* -> **Apply**.
6. *Select* -> **OK**, to confirm that the application settings have been successfully saved.
7. *Select* -> **OK**, to close the dialog and continue.

Processing Tab

Process

The *Process* feature can be used to manually run the volume calculation after manual loading of a dataset (Figure 93). To configure the manual processing of a dataset:

1. *Select* -> **Exclusion Boundaries** -> **Boundary**-> **Active** in the Project Tree (if required).
2. *Select* -> **Boundaries** -> **Boundary**-> **Active** in the Project Tree (if required).
3. *Select* -> **Floor** -> **Fixed Floor / Loaded Floor** -> **Active** in the Project Tree.
4. *Select* -> **Scans** -> *user defined file* to process from the Project Tree.
5. *Select* -> **Processing** tab -> **Process** icon, to start processing.
6. When the processing has been completed a dialog will be shown.
7. *Select* -> **OK**, to continue.

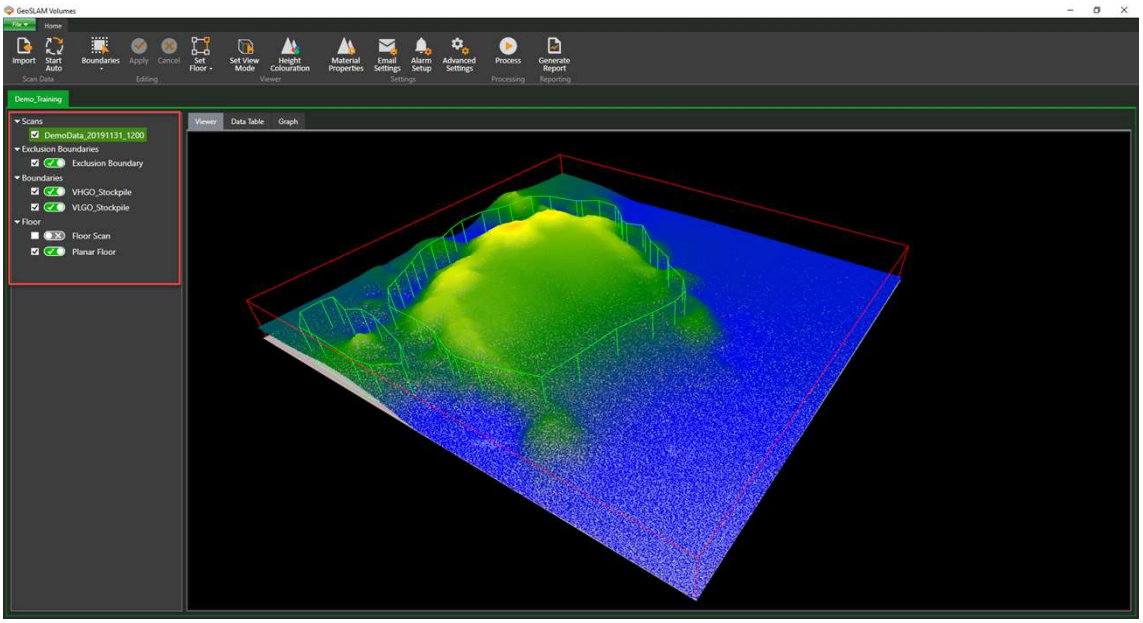


Figure 93. Manual processing, by activating the floor, boundary and selecting the data to process.

Once the processing of the data is complete, volumetric results will be displayed in the *Data Table*, either per project or per boundary, and in the *Graph* view, where data is displayed per boundary.

Reporting Tab

Generate Report

Selecting the *Generate Report* icon will open a secondary *Generate Report* Dialog (Figure 94).

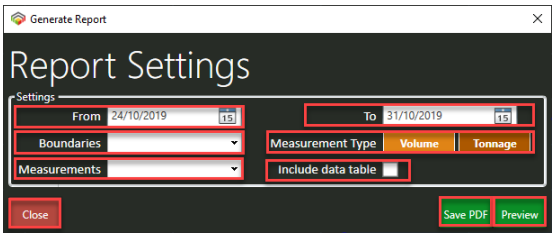


Figure 94. *Generate Report* settings dialog.

To create a report:

1. *Select* -> **Reporting** tab -> **Generate Report** icon.
2. This will open the *Generate Report* dialog (Figure 94).
3. *Select* -> **Settings** -> **From** -> start date of report.
4. *Select* -> **Settings** -> **To** -> start date of report.
5. *Select* -> **Boundaries** -> *Check* -> *boundary names* to include in the report.

6. *Select* -> **Measurements** -> *Check* -> *Cut*, *Fill* and / or *Net* values to report.
7. *Select* -> **Measurement Type** -> *Volume or Tonnage* values to report.
8. *Select* -> **Include data table** -> **CHECKED** -> to include the data table in the report.
9. *Select* -> **Preview**, to view a preview of the report (Figure 95). The report will include as a minimum:
   - The reporting period.
   - The boundaries reported.
   - For each boundary, the following parameters are reported:
     - Boundary Area.
     - Specific Gravity.
     - Cut Bulking Factor
     - Fill Bulking Factor.
   - Screenshot of the graph covering the report time period.

10. *Select* -> **Save PDF,** if the user parameters are acceptable:
11. *Browse* -> user folder to save the report.
12. Select -> **Save**, to continue.
13. Once the report has been created a *Generate Report* confirmation dialog will be displayed.
14. *Select* -> **OK**, to continue.
15. *Select* -> **Close**, to close the dialog and continue.

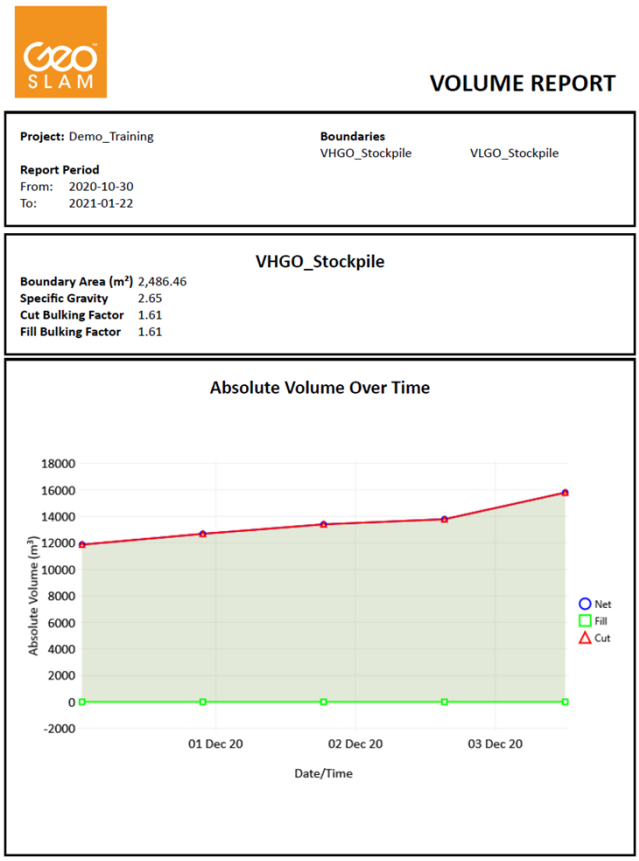


Figure 95. Example Volume Report.

Panel 3 – Project Information Bar

GeoSLAM VOLUMES allows the user to open multiple projects within interface. As the user opens or creates new projects, a new project tab will be visible on the *Project Information* bar (Figure 96). This enables the user to quickly move between projects. The current project is highlighted in green.

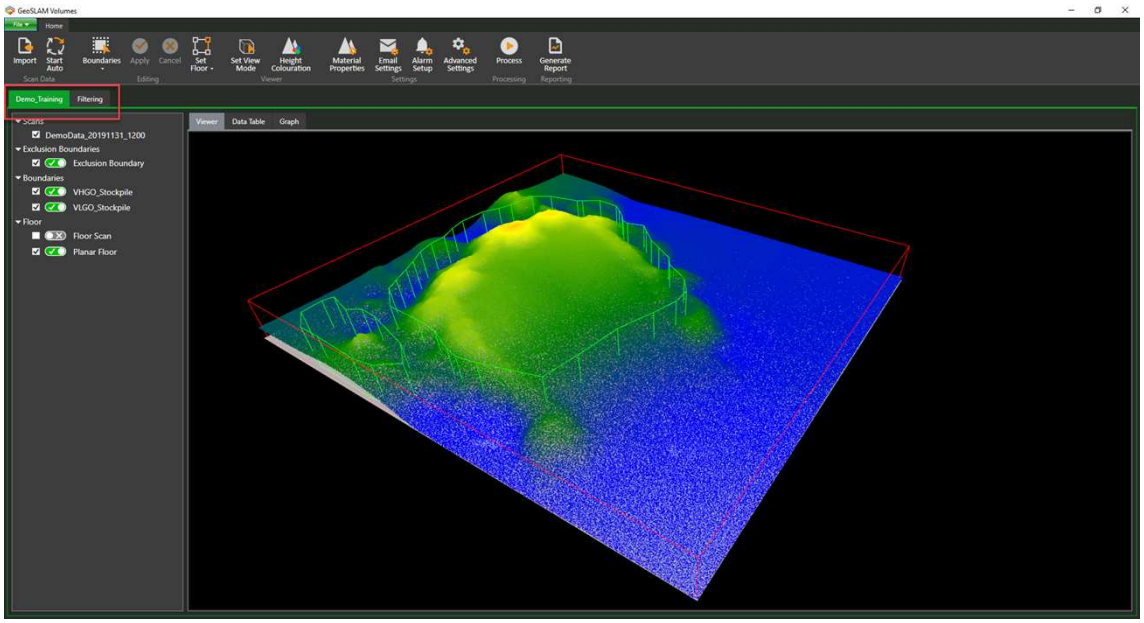


Figure 96. Project Panel allows the user to quickly navigate between multiple projects.

Panel 4 - Project Tree

The Project Tree is located to the left of the display window (Figure 97) and displays all layers and data currently included in the project. A description of the Project Tree items is given in Table 13.

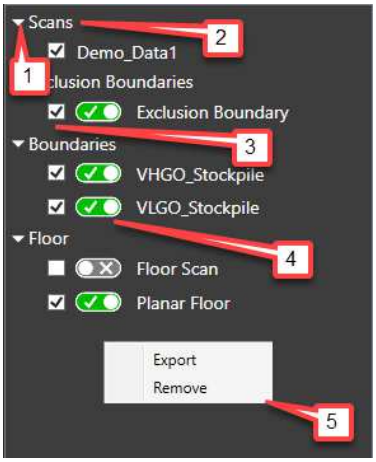


Figure 97. Project Tree.

Item	Item	Description
1	Layer Groups	There are several pre-set *Layer Groups* that will contain *Layers*. • Scans: contains list of the currently loaded data files. • Exclusion Boundaries: contains the exclusion boundary. • Boundaries: contains all created boundary area layers. • Floor: contains the Floor layers.
2	Expand/Collapse Button	Expands or collapses the corresponding layer group so more items can be viewed.
3	Hide/Show Layer Check Box	Turns on / off display of the corresponding *Layer Group* or *Layer*.
4	Active Check Box	This checkbox determined whether the chosen item is included in any processing.
5	*Right-Click* Options	Using the *Right-Click* on a specific scan file, boundary or floor will open a secondary dialog, enabling the user to: • Scans: remove the scan from the project. • Boundaries: i) Remove the file from the project. ii) export the boundary to a DXF file. • Floor: remove the floor from the project.

Table 13. Description of the Legend items.

Panel 5 – Display Tabs

The *Display* tabs (Figure 98) allow the user to switch between the:

a) **Viewer** tab -> Allows the user to view the imported pointcloud.
b) **Data Table** tab -> Allows the user to view the volumetric analysis data in a tabular format.
c) **Graph** tab -> Allows the user to view the volumetric analysis data in a graphic format over specific time periods.

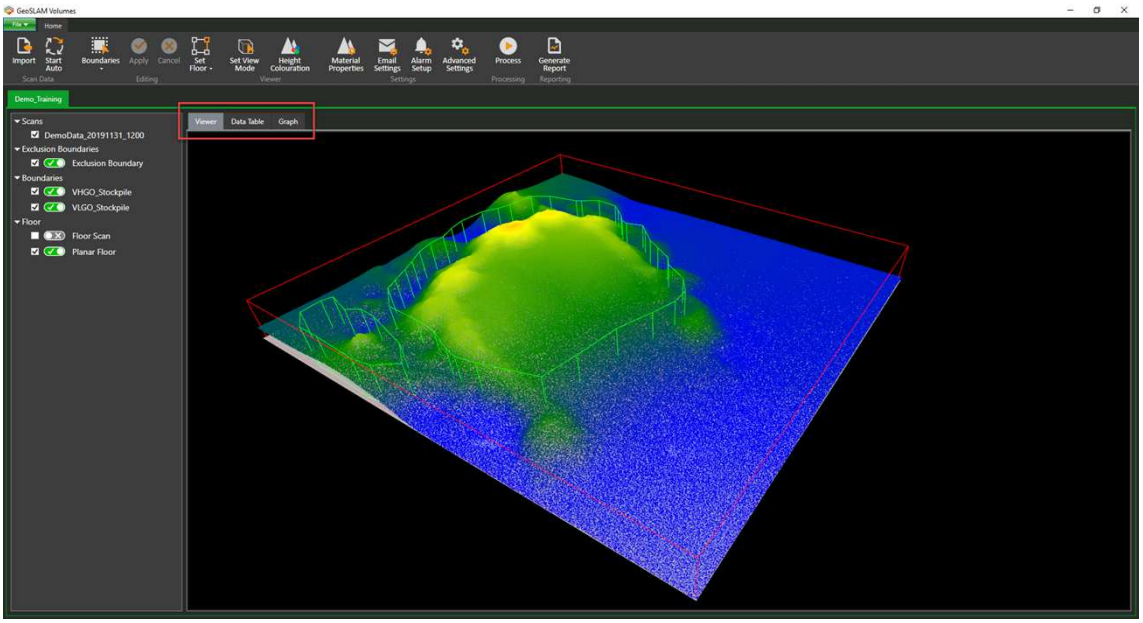


Figure 98. Display tabs allow the user to switch between the Viewer, Data Table and Graph views.

Viewer Tab

The *Viewer* window provides the user with a display of the currently selected data file. Navigation in the viewer window is described in Table 14.

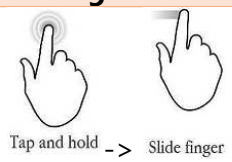
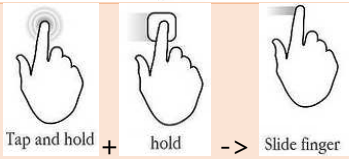
Function	Touchscreen	Mouse
	Navigation Mode	
Rotate		Hold down the left mouse button and dragging will rotate the scan.
Zoom In		Mouse Wheel Fwd.
Zoom Out		Mouse Wheel Rev.
Pan		Holding down the right button and dragging will translate the scan position in space.

Table 14. Data navigation in the view window.

Data Table Tab

All processed volumetric data metrics are displayed in the *Data Table* (Figure 99). There a several options for which data to display in *Data Table*:

1. Selecting to use the *Global Project* to provide a single Volume and Tonnage per dataset.
2. Selecting to use the *Active Boundary* list to provide individual Volume and Tonnage per boundary per dataset. The user also has the option to filter these results by boundary.

Independent of the data source type, the user can filter the results by date.

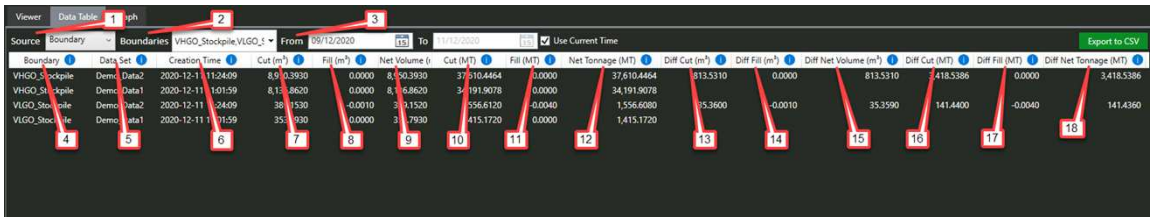


Figure 99. Example Data Table.

The Data Table is described in Table 15.

No.	Item	Description
1	Source	Sets whether the user wishes to see volumetric data based on a project level or at a boundary level.
2	Boundaries	Enables the user to select which boundaries are shown in the table.
3	From: To:	Enables the users to filter the results in the data table by specific date. *Select* -> Use Current Time to ensure that all the latest data is displayed in the table.
4	Boundary	Displays the boundary name that the data represents. *Note: Only available when the source is set to Boundary.*
5	Data Set	Display which dataset the data represents.
6	Creation Time	The time that the data file was created.
7	Cut (m^3)	Volume of material above the reference floor (V_C) (+ve).
8	Fill (m^3)	Volume of material below the reference floor (V_F) (-ve).
9	Net Volume (m^3)	Net Volume ($V_N = V_C + V_F$).
10	Cut (MT)	Tonnage of material above the reference floor (T_C) (+ve).
11	Fill (MT)	Tonnage of material below the reference floor (T_F) (-ve).
12	Net Tonnage (MT)	Net Tonnage ($T_N = T_C + T_F$).
13	Delta Cut (m^3)	Difference in the Cut Volume between the latest dataset and the previous dataset.

14	Delta Fill (m^3)	Difference in the Fill Volume between the latest dataset and the previous dataset.
15	Delta Net Volume (m^3)	Difference in the Net Volume between the latest dataset and the previous dataset.
16	Delta Cut (MT)	Difference in the Cut Tonnage between the latest dataset and the previous dataset.
17	Delta Fill (MT)	Difference in the Fill Tonnage between the latest dataset and the previous dataset.
18	Delta Net Tonnage (MT)	Difference in the Net Tonnage between the latest dataset and the previous dataset.

Table 15. Description of data table.

Viewing Global Parameters.

To view the processed data metrics for the global project:

1. *Select* -> **Source** dropdown -> **Project**.
2. To filter by date:
    a. *Select* -> **Calendar** icon, to choose the start date.
    b. *Select* -> **Calendar** icon, to choose the end date or *Check* -> **Use Current Time** to display data including the latest scan.
3. The Data Table will automatically update with the specified data (Figure 100).

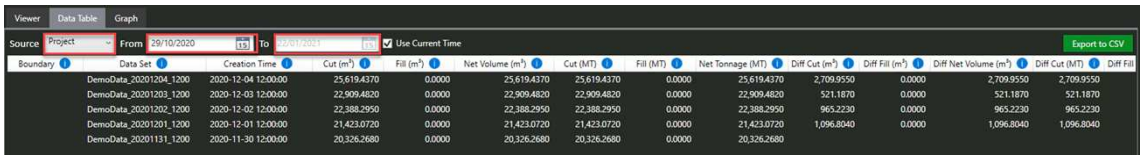
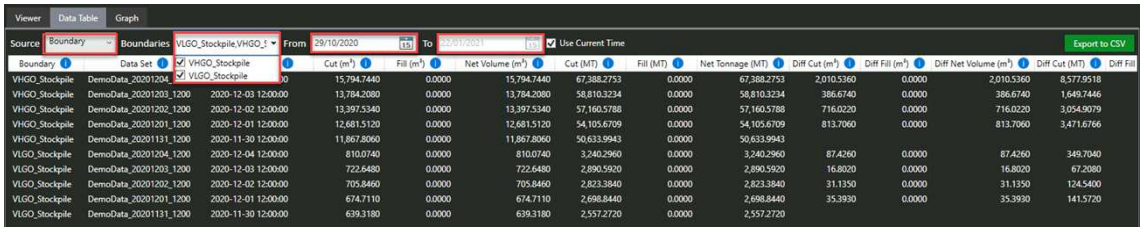


Figure 100. Example volumetric data table using the project source.

Using Boundary Parameters

To view the processed data metrics for by individual boundary:

1. *Select* -> **Source** dropdown box -> **Boundary**.
2. *Check* -> **Boundaries** dropdown box -> *Active Boundary* to display.
3. To filter by date:
    a. *Select* -> **Calendar** icon, to choose the start date.
    b. *Select* -> **Calendar** icon, to choose the end date or *Check* -> Current time to display the latest scans.
4. The Data Table will automatically update with the specified data (Figure 101).



Boundary	Data Set		Cut (m³)	Fill (m³)	Net Volume (m³)	Cut (MT)	Fill (MT)	Net Tonnage (MT)	Diff Cut (m³)	Diff Fill (m³)	Diff Net Volume (m³)	Diff Cut (MT)	Diff Fill
VHGO_Stockpile	DemoData_20201204_		15,794.7440	0.0000	15,794.7440	67,388.2753	0.0000	67,388.2753	2,010.5360	0.0000	2,010.5360	8,577.9518	
VHGO_Stockpile	DemoData_20201203_1200	2020-12-03 12:00:00	13,784.2080	0.0000	13,784.2080	58,810.3234	0.0000	58,810.3234	386.6740	0.0000	386.6740	1,649.7446	
VHGO_Stockpile	DemoData_20201202_1200	2020-12-02 12:00:00	13,397.5340	0.0000	13,397.5340	57,160.5788	0.0000	57,160.5788	716.0220	0.0000	716.0220	3,054.9079	
VHGO_Stockpile	DemoData_20201201_1200	2020-12-01 12:00:00	12,681.5120	0.0000	12,681.5120	54,105.6709	0.0000	54,105.6709	813.7060	0.0000	813.7060	3,471.6766	
VHGO_Stockpile	DemoData_20201131_1200	2020-11-30 12:00:00	11,867.8060	0.0000	11,867.8060	50,633.9943	0.0000	50,633.9943					
VLGO_Stockpile	DemoData_20201204_1200	2020-12-04 12:00:00	810.0740	0.0000	810.0740	3,240.2960	0.0000	3,240.2960	87.4260	0.0000	87.4260	349.7040	
VLGO_Stockpile	DemoData_20201203_1200	2020-12-03 12:00:00	722.6480	0.0000	722.6480	2,890.5920	0.0000	2,890.5920	16.8020	0.0000	16.8020	67.2080	
VLGO_Stockpile	DemoData_20201202_1200	2020-12-02 12:00:00	705.8460	0.0000	705.8460	2,823.3840	0.0000	2,823.3840	31.1350	0.0000	31.1350	124.5400	
VLGO_Stockpile	DemoData_20201201_1200	2020-12-01 12:00:00	674.7110	0.0000	674.7110	2,698.8440	0.0000	2,698.8440	35.3930	0.0000	35.3930	141.5720	
VLGO_Stockpile	DemoData_20201131_1200	2020-11-30 12:00:00	639.3180	0.0000	639.3180	2,557.2720	0.0000	2,557.2720					

Figure 101. Example volumetric data table using the boundary source.

Data Export

Choosing either to view the data using the global or boundary option, the user has the ability to export the currently viewed data table to a standard comma delimited (*.CSV) file format.

To export the date to a CVS file:

1. *Select* -> **Data Table** -> **Export to CSV**.
2. *Browse -> required output folder.*
3. *Input -> user defined filename.*
4. *Select* -> ***Save***.
5. *Select* -> **OK**, at the *Export to CSV* dialog to continue.

Graph Tab

All volumetric data metrics relating to the user defined boundaries can be displayed in the graphic tab that provides a graphical display of either the Volume or Tonnage data (Figure 102).

Figure 102. Graph view displaying volume data.

The Graph window options are described in Table 16 with navigation in the Graph area described in Table 17.

No.	Item	Description
1	Date Filters	The user has the ability to view data from the current *Day*; *Week*; *Month* or to set a selected data range using the *From*: and *To*: calendars.
2	Settings	The user can select to view either Volume data or Tonnage data over the selected period. With either setting, the user can select to view the Cut, Fill or Net values individually or together.
3	Boundaries	Any user defined boundaries created in the project will be displayed. Note: Only one boundary can be displayed at one time.
4	Graph	This is the main graph window. It will display either the Volume or Tonnage over time depending on the settings.
5	Timescale	The graph can be Zoom-in and Out using the mouse wheel option to display more or less time. The user can then scroll the zoomed timescale.

Table 16. Description of Graph View.

Function	Touchscreen	Mouse
Navigation Mode		
Scroll	N/A	Hold down the left mouse button and dragging will scroll the view.
Zoom In	N/A	Mouse Wheel Fwd.
Zoom Out	N/A	Mouse Wheel Rev.

Table 17. Navigation in the Graph window.

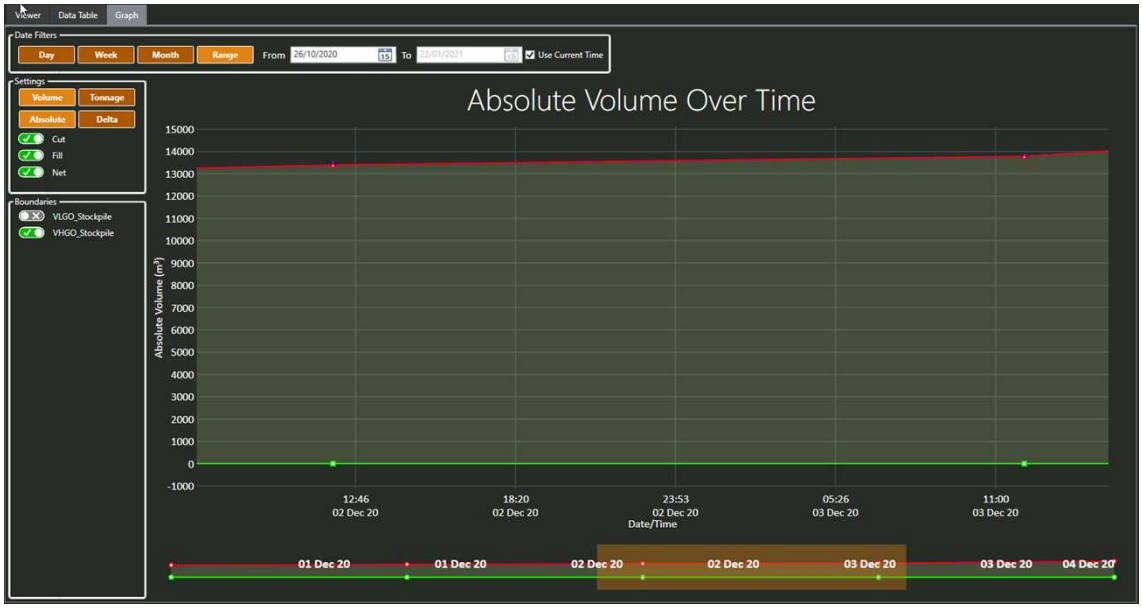


Figure 103. Expanded time series graph.

To view the data graphically in the Graph view:

1. *Select* -> **Graph** tab.
2. *Select* -> Data Filters ->
    a. *Select* -> either **Day**, **Week**, **Month**, or
    b. *Select* -> **Range** ->
        i. *Select* -> **Calendar** icon, to choose the start date.
        ii. *Select* -> **Calendar** icon, to choose the end date or *Check* -> **Use Current Time** to display the latest scans.
3. *Select* -> **Settings** ->
    a. **Volume**, or
    b. Tonnage.
4. *Select* -> **Settings** ->
    a. **Cut,** and / or
    b. **Fill,** and / or
    c. **Net**.
5. *Select* -> **Boundaries** -> *boundary name* -> *Boolean* -> **Active**.

Part 5 – Technical Background.

Cut and Fill Nomenclature

Volume

3 volumetric calculations are undertaken in *GeoSLAM Volumes*. These are:

- The Cut volume (V_C): This is the volume between the reference floor and the 3D terrain (represented by the scan) when the terrain is higher than the reference floor.
- The Fill volume (V_F): This is the volume between the reference floor and the 3D terrain when the terrain is lower than the reference floor.

  Note: The Fill volume is given with negative value / sign.

- Net Volume (V_T): This is total volume and is calculated as:

$$\text{Total Volume} = V_T = V_C + V_F$$

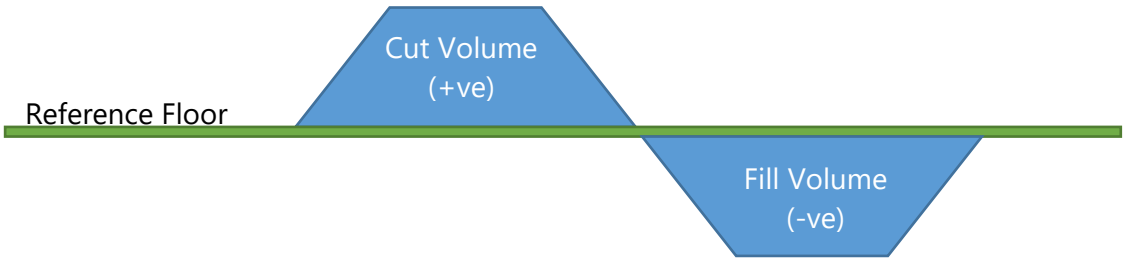


Figure 104. Definition of Cut and Fill Volume.

Tonnage

The standard relationship between tonnage and volume can be described as:

$$\text{Tonnage (kg)} = \text{Volume (m}^3\text{)} * \text{Bulk Density (kg/m}^3\text{)}$$

Where the bulk density of the material in the stockpile is not explicitly known it is possible to estimate it by utilising the known specific gravity of the primary mineral and using the following method.

$$\text{Density } (_{\text{material}}) = \text{specific gravity}_{(\text{material})} \text{ x specific gravity}_{(\text{water})}$$

This allows the calculation of the material density. However, within the stockpile, there will be a significant number of voids introduced into the material due to the extraction process. Therefore, we must allow for this additional void space to calculate an effective bulk density, where:

$$\text{Effective Bulk Density} = \text{Density}(_{\text{material}}) \text{ x Bulking Factor.}$$

Where the bulking factor is expressed as a ratio.

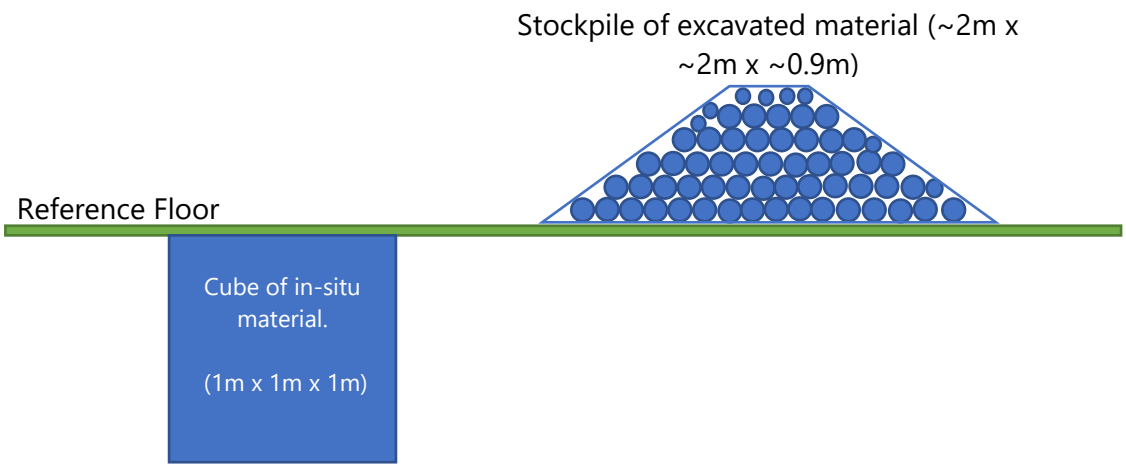


Figure 105. Bulking Factor.

If the user knows the bulk density of the material on the stockpile then when entering values into the Set Material Properties dialog, it is not necessary to account for the bulking factor.

e.g., if the user knows the bulk density of the material on the stockpile is 4372.5 kg/m^3.

- *Input* -> **Specific Gravity** = 4.3725.
- *Input* -> **Cut Bulk Factor** = 1.
- *Input* -> **Fill Bulk Factor** = 1.

The software will then display the Cut Bulk Density = 4372.50.