User Manual

Software version: 1.1

Table of Contents

Table of Contents

Table of Contents

1 Introduction

1.1 About HiveMap

HiveMap is a cutting-edge mapping solution that empowers geoscience professionals to capture high-quality data in the field or office and relay this info into important geological and geotechnical modelling and analysis.

HiveMap was developed by SRK Consulting to support improved mapping and data interpretation of remotely collected rock exposure data. This was motivated by a desire to increase data availability to support improved geological interpretations and products for their clients. SRK remains intimately involved in the direction of HiveMap, with a team of experienced industry professionals supporting its development and engagement with users.

- Precise Mapping
    - HiveMap delivers high-resolution, precise geological mapping that ensures reliable data for better decision-making.
- Easily Integrates
    - Integrate HiveMap with existing datasets and downstream modelling packages for a streamlined workflow.
- Collaborate in Real-Time
    - Foster real-time, safe, cross-functional collaboration among geologists and engineers, regardless of location.
- Quick to Generate
    - Capture high-confidence geological observations that are quickly and immediately usable for analysis, interpretation, and discussion.

1.2 What Is Mapping?

Mapping is the process of looking at the rocks and recording what we see. Depending on our technical discipline, what we record, and the level of detail achieved can vary significantly. It is also important to understand that mapping involves recording the location of geological features, their orientations, and their attributes. These fundamental observations must be recorded properly and consistently to support downstream analysis and modeling work.

To learn more about HiveMap, check out our blog posts: https://hivemap3d.com/blog/

1.3 Software Installation

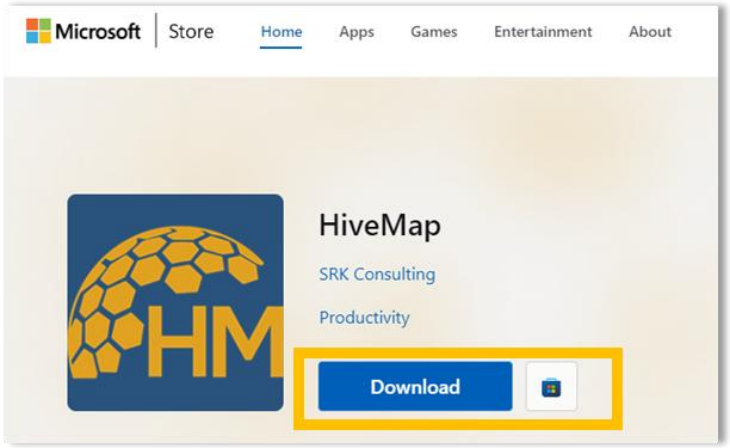
There are two ways to download HiveMap:

1. Using the Microsoft Store app
2. Direct download provided by the HiveMap Team (to be used if there are company IT restrictions on use of Microsoft Store).

1.4 Microsoft Store

To download HiveMap, install the program from the Microsoft store & follow the install instructions: https://apps.microsoft.com/detail/9mvg8cv6n1q4

Microsoft store download link.



Click Download, and if prompted, click Open Microsoft Store to get to the download page.

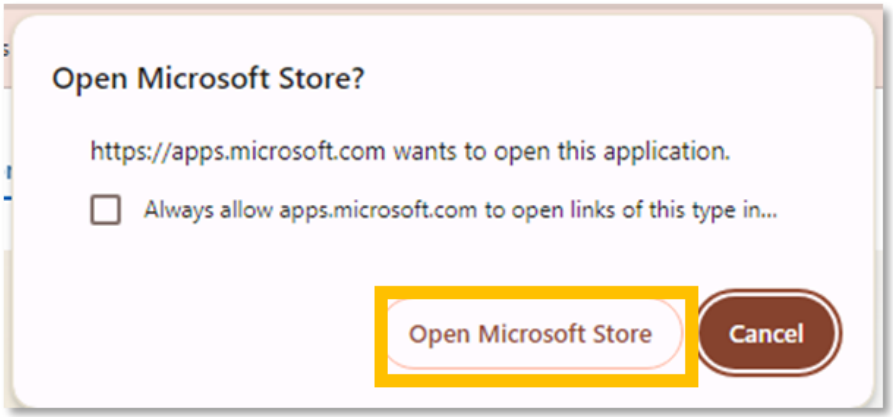


Table of Contents

Click on Get from the Microsoft store pop up.

HiveMap will begin to download. A desktop notification will pop up once installation is completed.

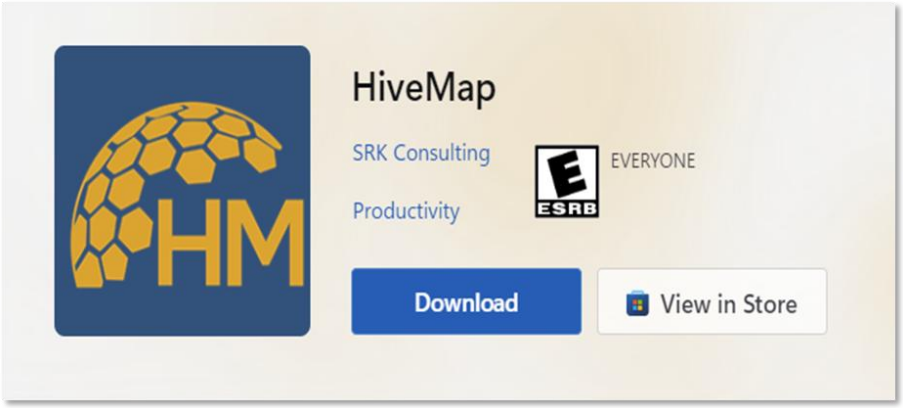


Table of Contents

1.5 Direct Download

If there are any issues with the system installation through Microsoft Store, please submit an IT Support Request Form. Help Desk will follow up and provide a zipped folder for direct download of the software.

2 Quick Start Guide

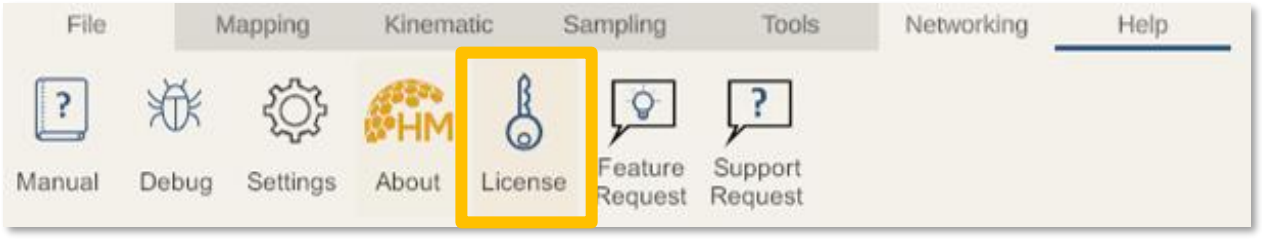
In this section an overview of HiveMap features are outlined in a tutorial format. See User Guide for more information (Section 3.0).

2.1 License Registration

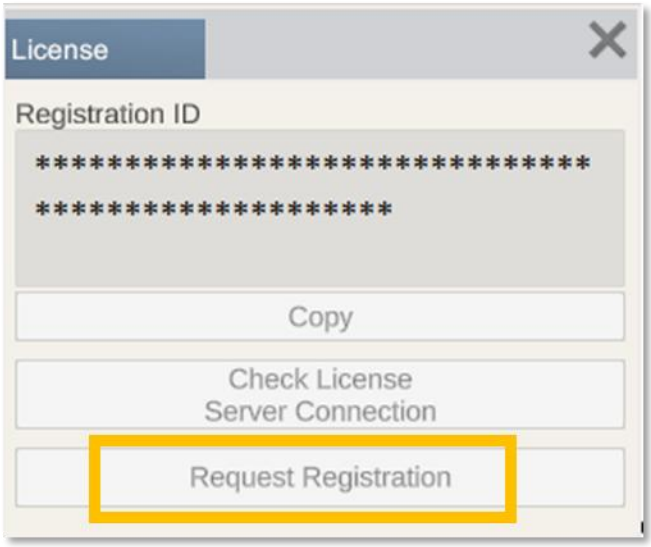
Once HiveMap has been installed, open the software to register the program license. Take the following steps:

Step 2.1-1:Click Help in the navigation bar at the top of the screen. Choose the 'License' button.
Step 2.1-2:On the License popup window, click on the Request Registration button which will open the license registration form.
Help Desk will process your licensing request once the form is received.
Users will receive an email from Help Desk once the license is confirmed set up and active.
To activate the user license, restart HiveMap.

Step 2.1-1: Click Help in the navigation bar at the top of the screen. Choose the 'License' button.



Step 2.1-2: On the License popup window, click on the Request Registration button which will open the license registration form.



2.2 System Set Up

Before you can begin creating projects in HiveMap, the system settings need to be configured with a Username and a file path location for the HiveMap main folder, where all project files will be saved and accessed from.

Step 2.2-1: To set up HiveMap, go to Settings (top right corner, gear symbol) and update the system settings.

Step 2.2-2: The username and the main file location should then be updated.

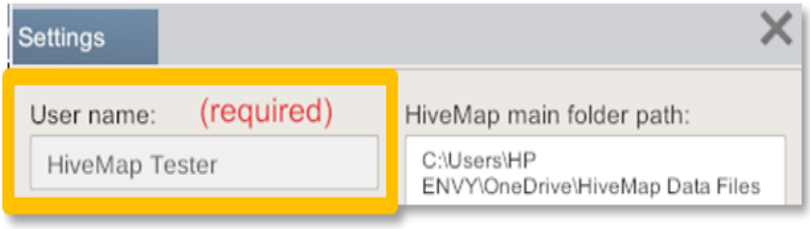
Step 2.2-3: To update the main folder location, click on Select Main Folder.

Step 2.2-4: Once the username and main folder path location are updated, click Apply & Save Settings**.**

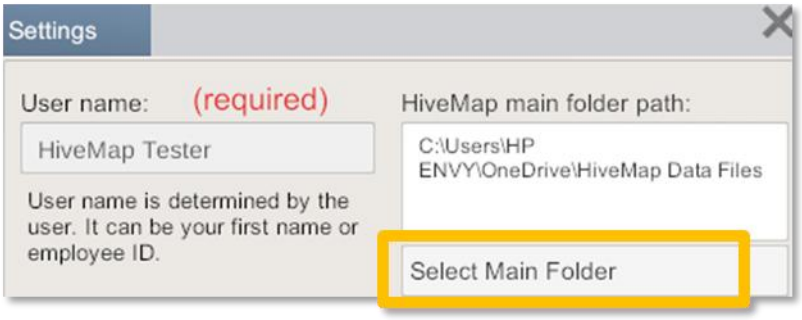
Table of Contents

Step 2.2-1: To set up HiveMap, go to Settings (top right corner, gear symbol) and update the system settings.

Step 2.2-2: The username and the main file location should then be updated.



Step 2.2-3: To update the main folder location, click on Select Main Folder.



Step 2.2-4: Once the username and main folder path location are updated, click Apply & Save Settings**.**

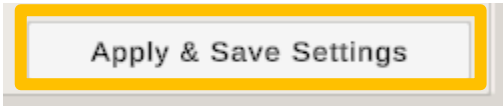


Table of Contents

2.3 User Interface

The HiveMap main window has 3 main areas: ribbon menu (red), view menu (blue), and workspace (green).
The ribbon menu is located on the top of the HiveMap window. It contains tabs which groups tools and options. Mapping (section 3.4), Kinematic (section 3.5), Sampling (section 3.6), Tools (section 3.7), Networking (section 3.8) and Help (section 3.9) tabs can be accessed from here. Ribbon menu can be hidden with Hide/Unhide ribbon button.
View menu is located left side of the window. View menu is used to adjust the camera of the 3D View and display settings of the objects in the 3D View (section 3.3).
Workspace has dockable panels that can be dragged interactively around the screen to make them float over any location, and dock back to panels. Below shows the default layout of the workspace.

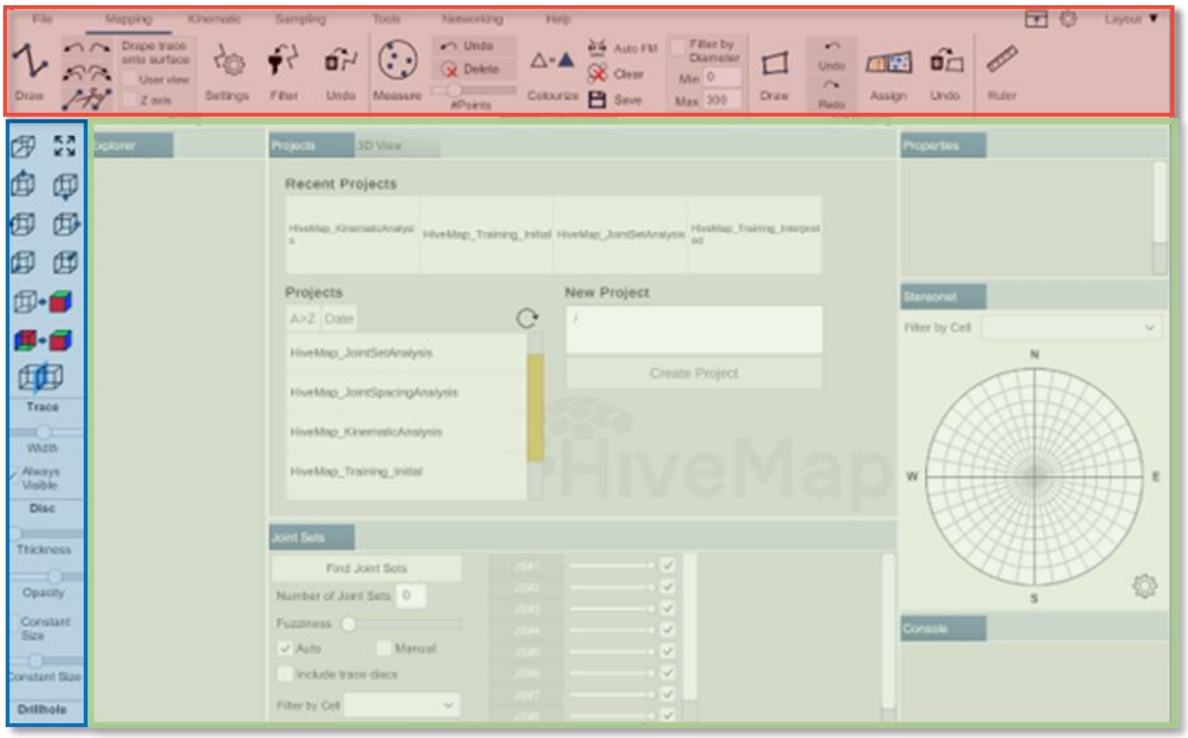


Table of Contents

There are 4 different default layouts, which include Default, Geotech, Geology, and Tablet. Users can switch between these layouts or create their own layout and save it. To save a custom layout, enter a name and click the Save button. To delete a custom layout, select the *X* beside the specific layout and it will be deleted.

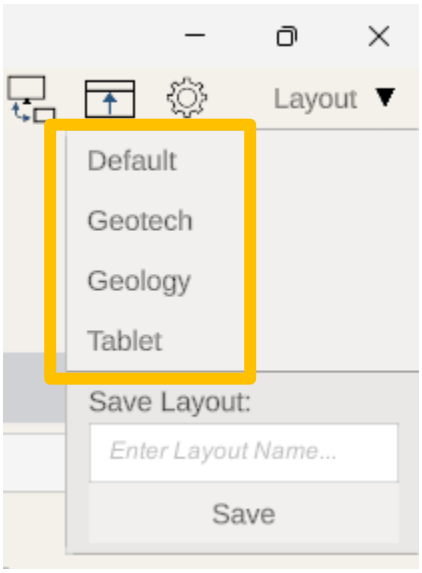
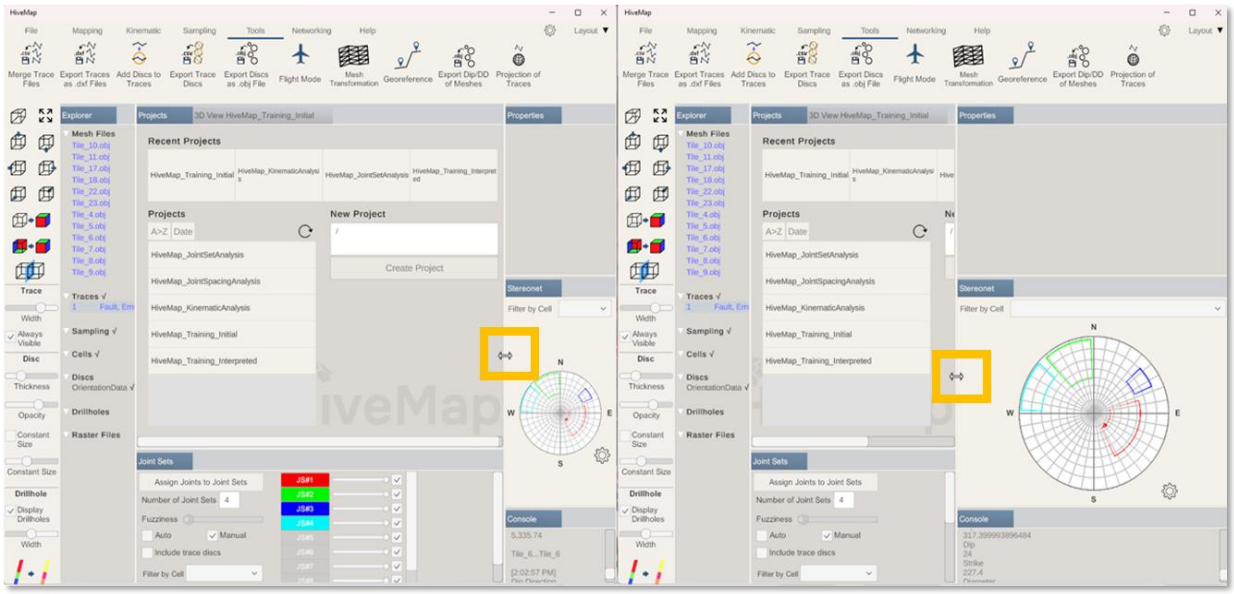


Table of Contents

2.4 Adjusting panel layout

All panel sizes can be adjusted in the workspace.

Move your cursor to the edge of the panel you want to resize. When the cursor changes to a double-sided arrow, press, and hold the left mouse button. Drag the mouse to adjust the panel to your desired size.



A panel can be moved out from its docked position and become a floating panel or can be docked to a new position.

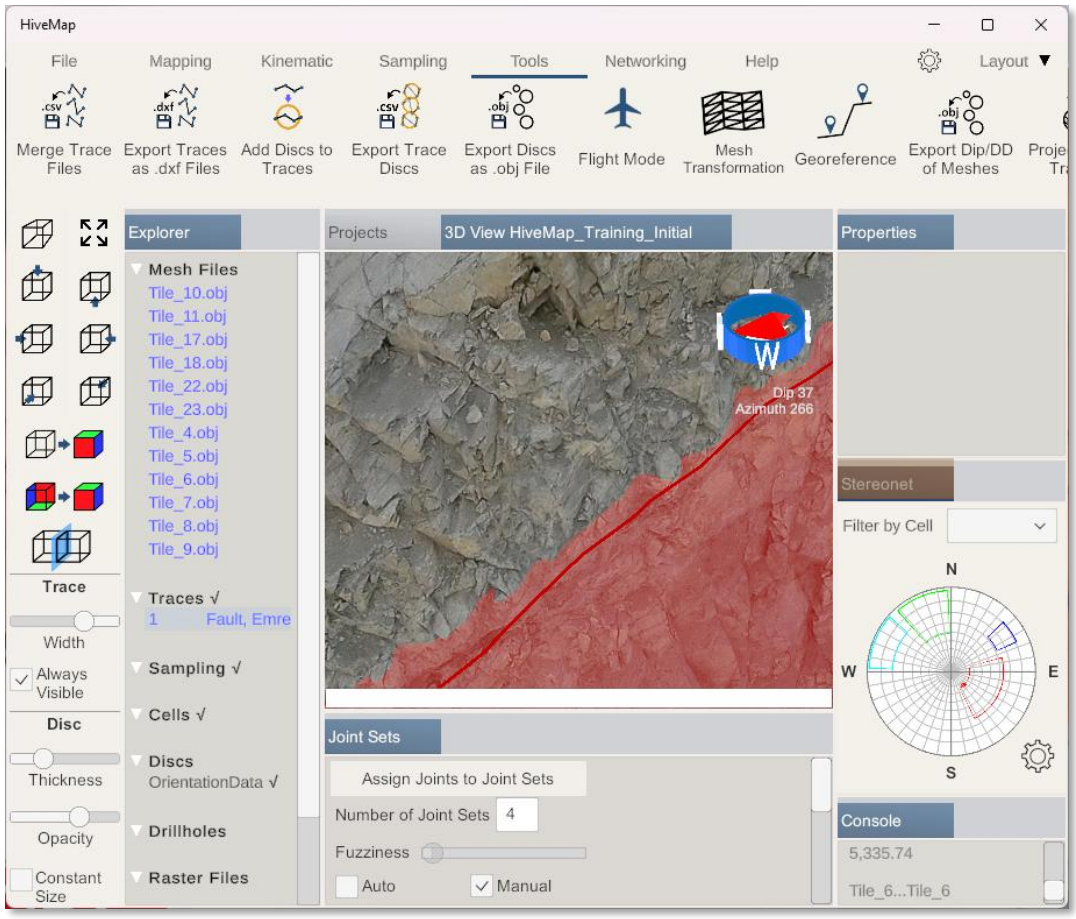
Step 1: Move your cursor to the title of the panel you want to move.

Step 2: Press and hold the left mouse button.

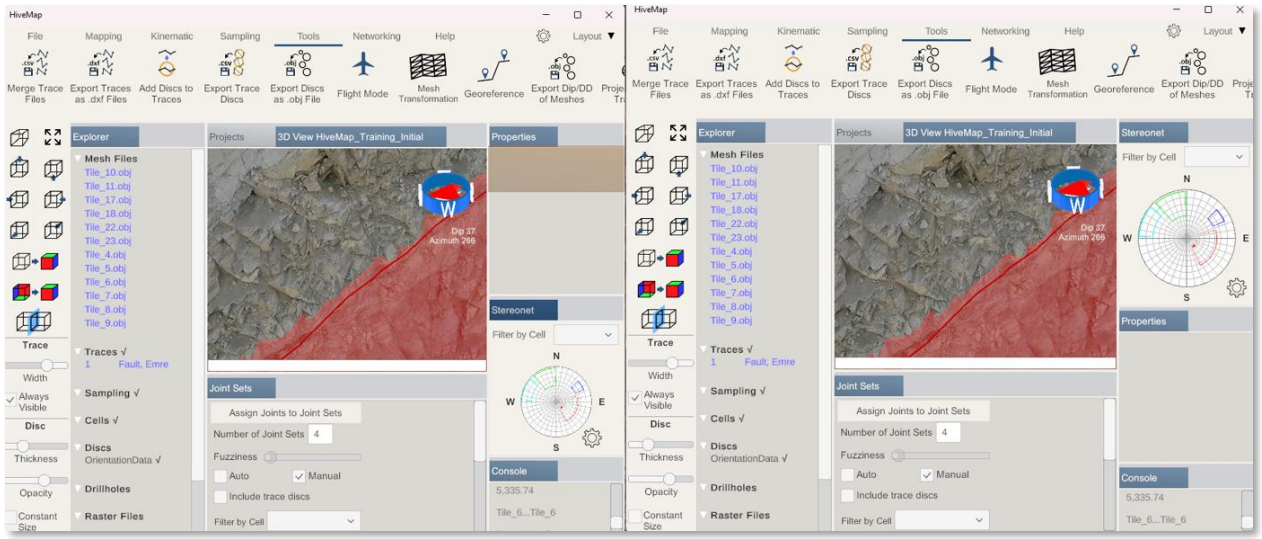
Step 3: Drag the mouse to move the panel. A shade of a panel will be displayed.

Step 4: Release the mouse button and the panel will appear where the panel shade was displayed.

Table of Contents

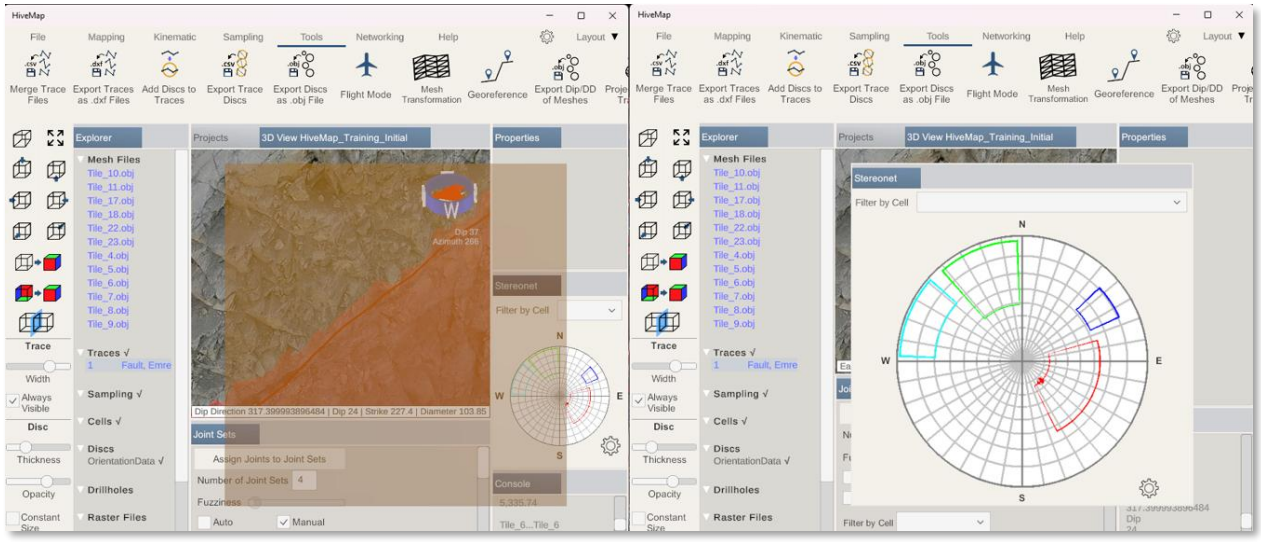


While dragging the panel, if the cursor gets close to a panel border in the workspace, the panel shade will appear docked to that border.

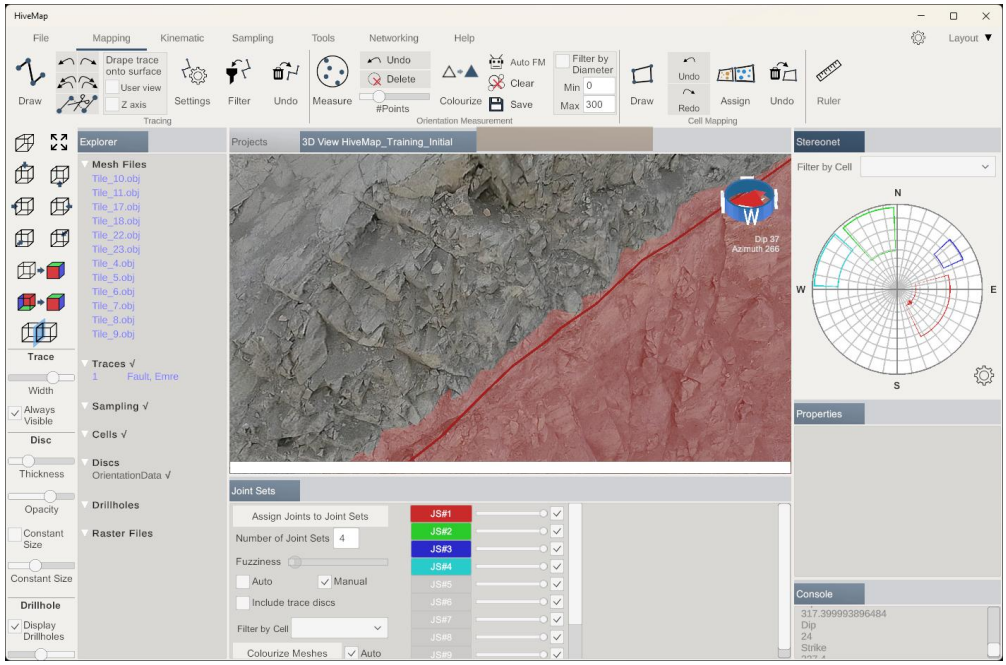


[Table of Contents]

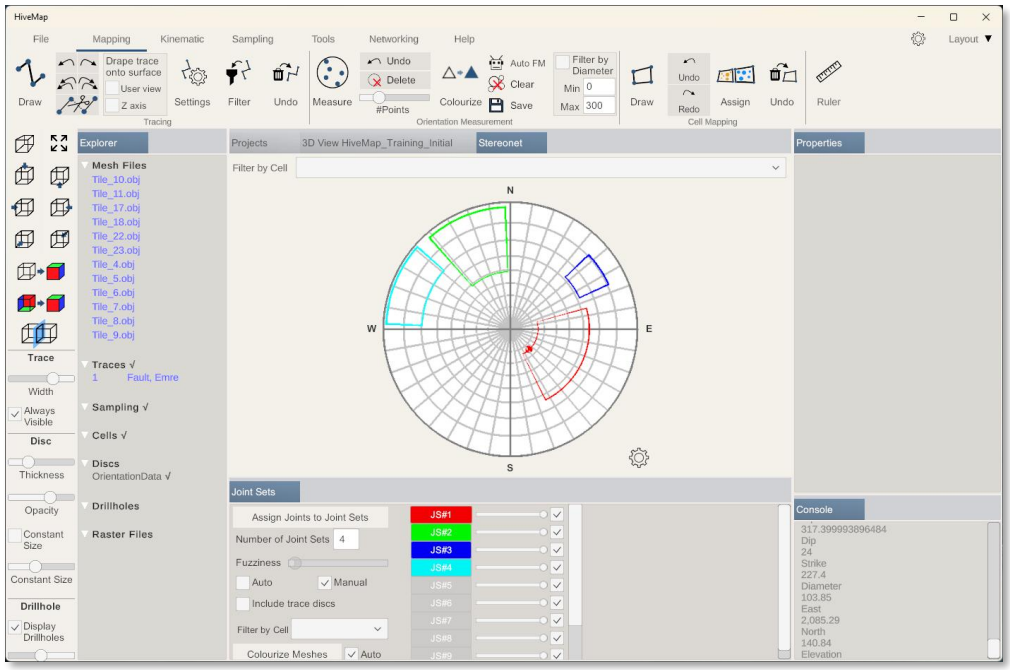
If it is not close to any panel border, the shade will appear as a floating panel.



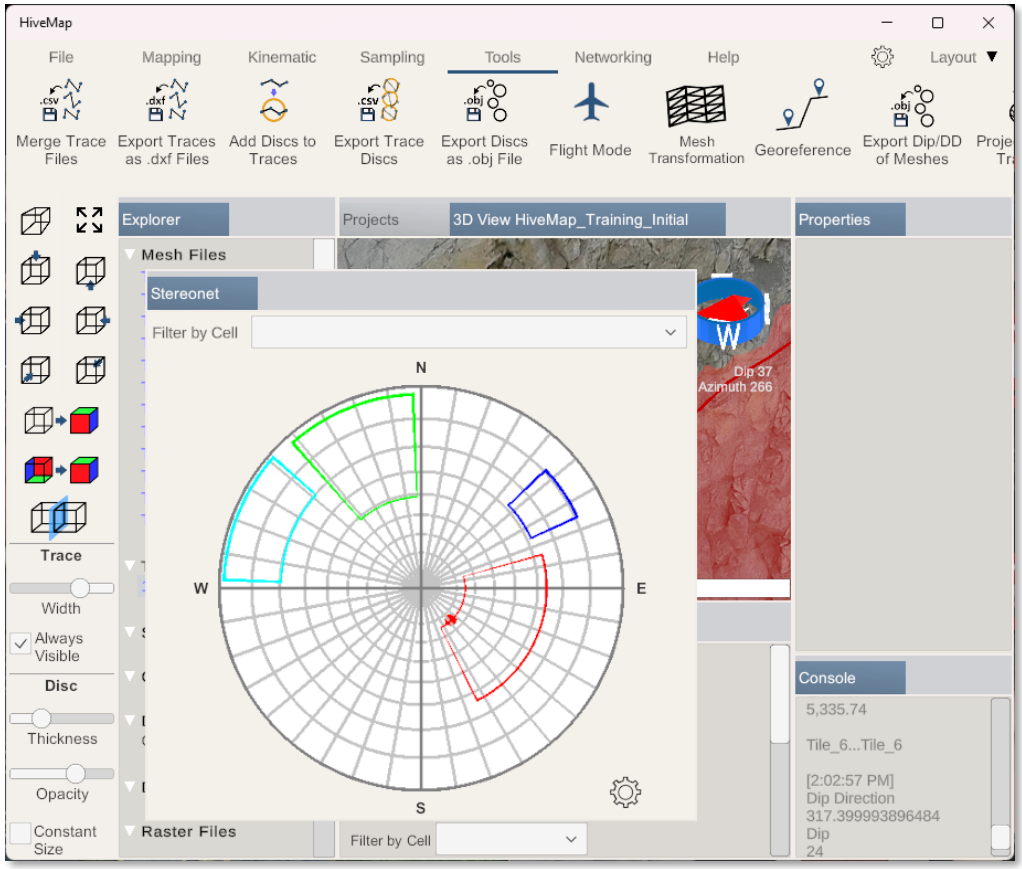
If the mouse cursor gets over a title bar of another panel, it will be attached as a tab to the panel.



[Table of Contents]



To move the floating panel, move your mouse cursor to the title bar, but not to the title. Press and hold the left mouse button. Drag the mouse to move the panel.



[Table of Contents]

2.5 Importing First Project

Users can access the HiveMap training data files on the website from the resources page: hivemap3d.com/resources.

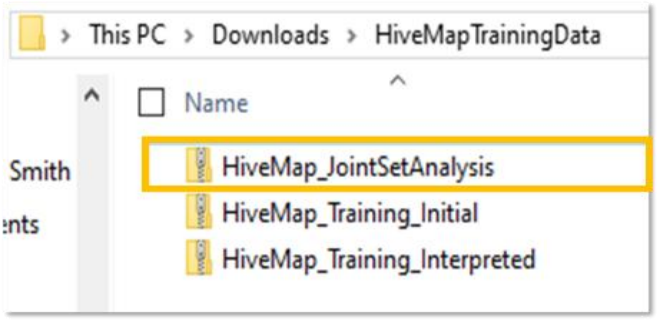
The HiveMap Training Data.zip file folder contains 3 zipped sub-folders as follows:

- HiveMap_JointSetAnalysis
- HiveMap_Training_Initial
- HiveMap_Training_Interpreted

Download the training data files zip folder, then extract all the files. There will be 3 zipped sub-folders within.

Step 2.5-1: Extract the files for each zipped sub folder.
Step 2.5-2: Copy these folders to the HiveMap main folder.
Step 2.5-3: Go to Projects tab.
Step 2.5-4: If the training data set is not visible on the Projects list, click the Refresh button.
Step 2.5-5: On the project list click “HiveMap_Training_Initial” project. All the data of this project will be listed in the Explorer panel. This project has 12 mesh files which are listed under the Mesh Files branch.
Step 2.5-6: Right click the “Mesh Files” branch which will select all the mesh files and display right click menu. Click the Load button.
Step 2.5-7: The loading status for each file will appear beside each file name.
Step 2.5-8: Once all the meshes have been loaded, the 3D View panel will automatically zoom in on the loaded mesh files.

Step 2.5-1: Extract the files for each zipped sub folder.



Step 2.5-2: Copy these folders to the HiveMap main folder.

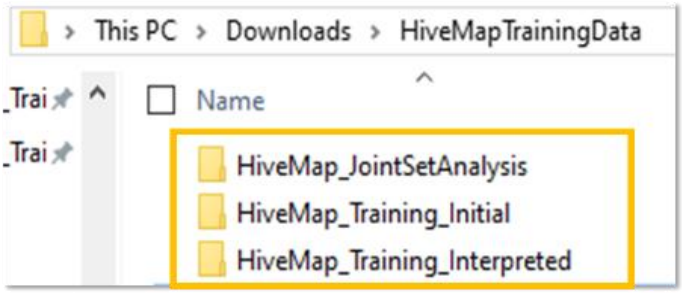
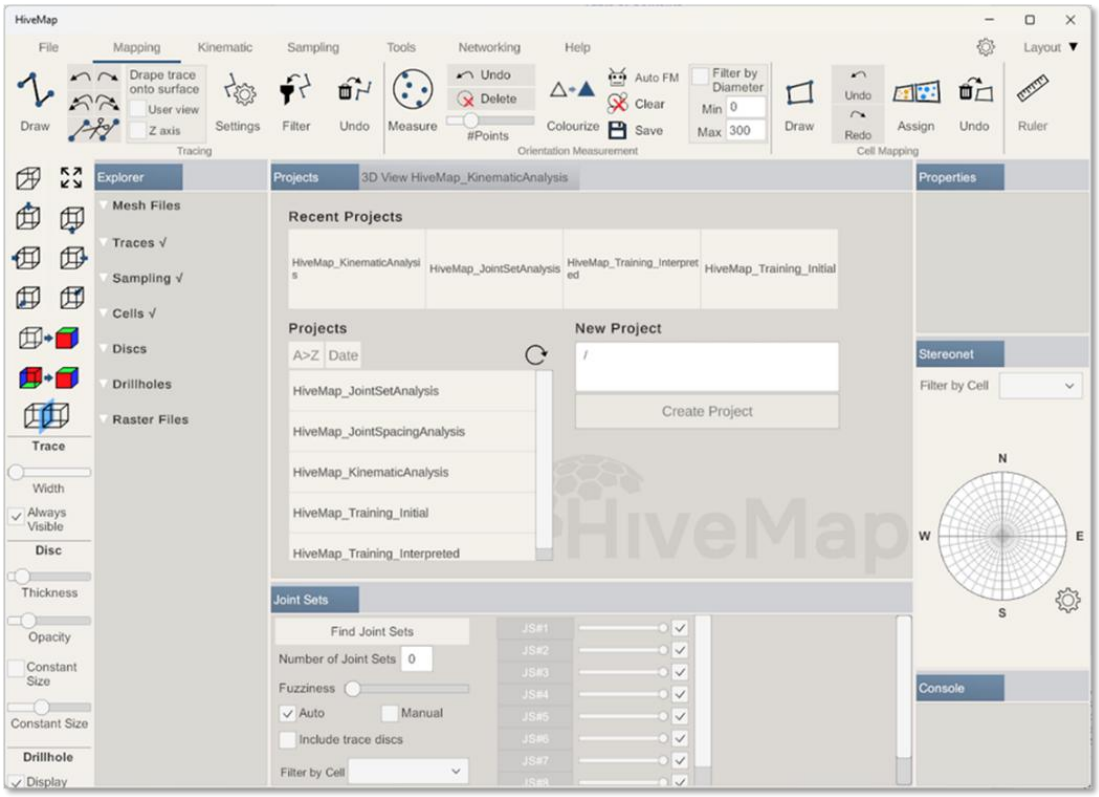
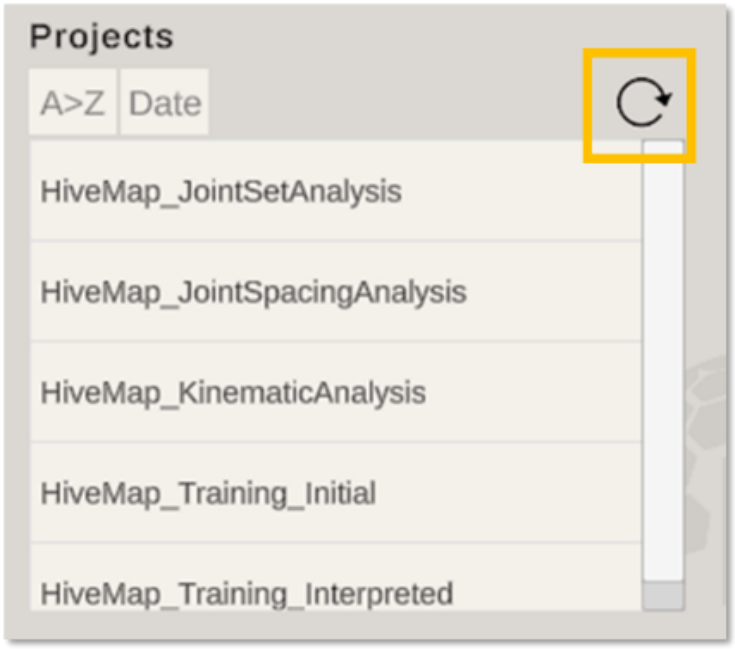


Table of Contents

Step 2.5-3: Go to Projects tab.



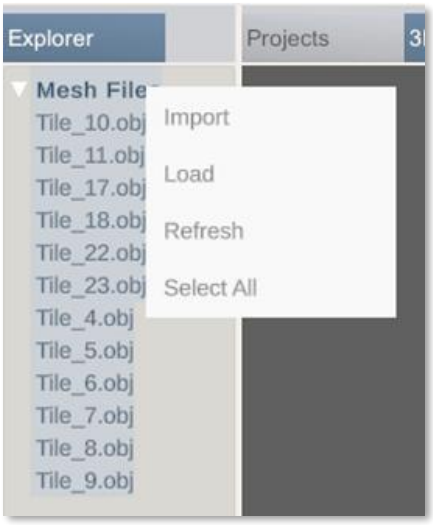
Step 2.5-4: If the training data set is not visible on the Projects list, click the Refresh button.



[Table of Contents]

Step 2.5-5: On the project list click “HiveMap_Training_Initial” project. All the data of this project will be listed in the Explorer panel. This project has 12 mesh files which are listed under the Mesh Files branch.

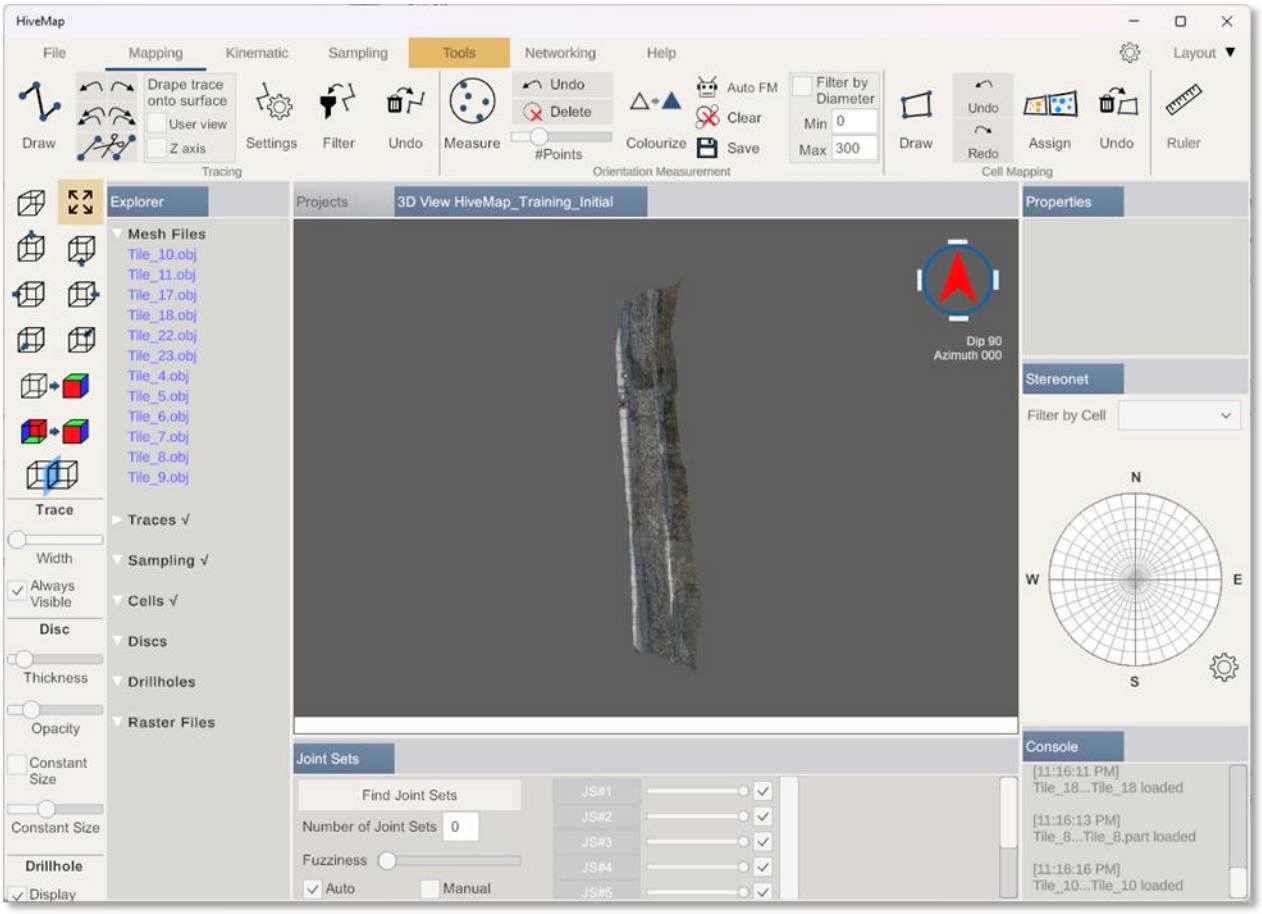
Step 2.5-6: Right click the “Mesh Files” branch which will select all the mesh files and display right click menu. Click the Load button.



[Table of Contents]

Step 2.5-7: The loading status for each file will appear beside each file name.

Step 2.5-8: Once all the meshes have been loaded, the 3D View panel will automatically zoom in on the loaded mesh files.



[Table of Contents]

2.6 Controls

To interact with the meshes:

- Use right mouse button to rotate the mesh.
- Use the scroll wheel on the mouse to zoom in and out.
- Use the middle mouse button (usually scroll wheel) to pan the mesh.

2.6.1 Hotkeys and shortcuts

A list of hotkeys and shortcuts for the software are included in the Table below.

Tracing	Enter or Right Click + Left Click	Stops the tracing and starts tracing with the same structure.
	Shift + Enter or Shift + Right Click + Left Click	Stops tracing
	Backspace	Undo last node
Tracing menu	Esc	Close trace editing menu without saving
	Shift + Delete	Delete the trace and close the menu
Orientation measurement	Enter or Right Click + Left Click	Create disc (If there is at least 3 sampling points)
	Backspace	Undo last sampling point
	Delete	Delete selected disc
Mesh colours	1	Texture
	2	Inverse colour
	3	Vertices colour
	4	Shade
	5	Wireframe
	6	Orientation
	7	Slope
	8	Aspect
	9	Elevation
Change view	Numpad 0	Switch between perspective and orthographic projection
	Numpad 5	Look down
	Numpad 2	Look south
	Numpad 6	Look west
	Numpad 8	Look north
	Numpad 4	Look east

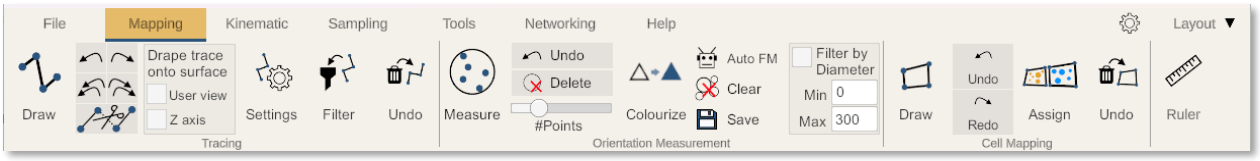
Table of Contents

2.7 Mapping Structures

Step 2.7-1: By using your mouse find a geological structure such as fault to trace on the mesh.
Step 2.7-2: Go to the Mapping tab on the ribbon and click on the Draw button on the Tracing group.
Step 2.7-3: A drop-down menu is displayed with the list of structures. Select one of these structures.
Step 2.7-4: Use your left mouse button to start tracing the structure on the mesh. You can use single clicks or hold down the left mouse button.
Step 2.7-5: After finishing tracing click on the Stop button to end the process.
Step 2.7-6: The new trace file is displayed on the Explorer panel under the Traces branch.
Step 2.7-7: To edit the trace file, left click the trace on the mesh. Alternatively, right click the trace on the Explorer panel and then the Edit Trace on the Right Click menu.
Step 2.7-8: The Trace Menu is displayed, and the nodes of the trace polyline/polygon becomes visible.
Step 2.7-9: Click the Add Point button and then click the trace line.
Step 2.7-10: A new node will be added where the line is clicked.
Step 2.7-11: Click one of the white nodes. The node will become red, and on the menu the Delete Point button will become visible. Click this button to delete the selected red node.
Step 2.7-12: Click the mesh and the red node will move to the clicked position on the mesh.
Step 2.7-13: Click the first or last node of the trace, which is coloured purple. The node will become red, and on the menu the Delete Point button and Continue Tracing button will become visible. Click Continue Tracing. This will close Trace Menu and hide the nodes and tracing will continue from the selected node.
Step 2.7-14: On the Trace Menu click the Calculate DD/Dip of Trace. A disc with the orientation of the structure will be created.
Step 2.7-15: To save the changes click X on the top right of the Trace Menu. A menu will be displayed.

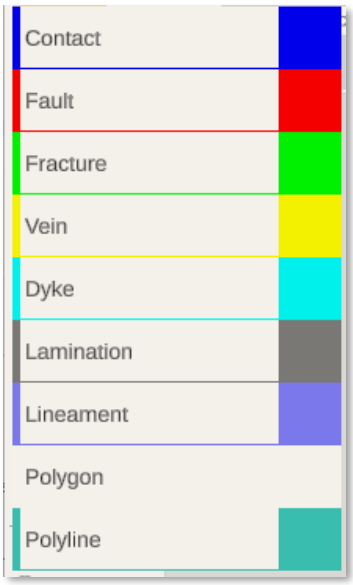
Step 2.7-1: By using your mouse find a geological structure such as fault to trace on the mesh.

Step 2.7-2: Go to the Mapping tab on the ribbon and click on the Draw button on the Tracing group.



[Table of Contents]

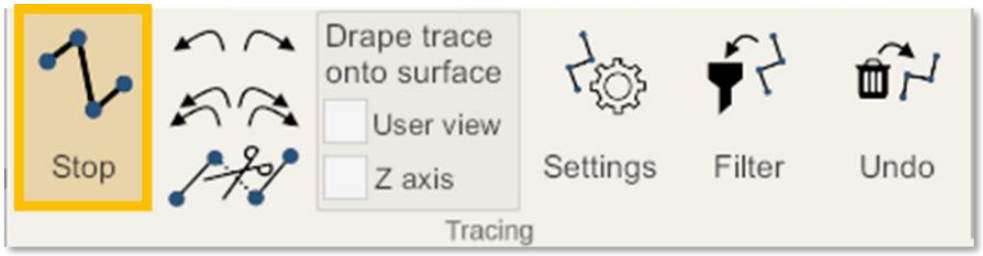
Step 2.7-3: A drop-down menu is displayed with the list of structures. Select one of these structures.



Step 2.7-4: Use your left mouse button to start tracing the structure on the mesh. You can use single clicks or hold down the left mouse button.

Table of Contents

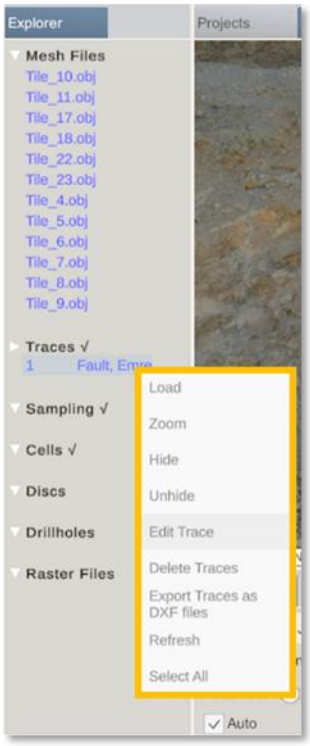
Step 2.7-5: After finishing tracing click on the Stop button to end the process.



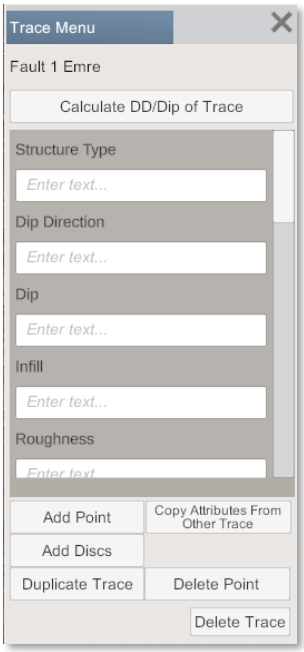
Step 2.7-6: The new trace file is displayed on the Explorer panel under the Traces branch.

[Table of Contents]

Step 2.7-7: To edit the trace file, left click the trace on the mesh. Alternatively, right click the trace on the Explorer panel and then the Edit Trace on the Right Click menu.

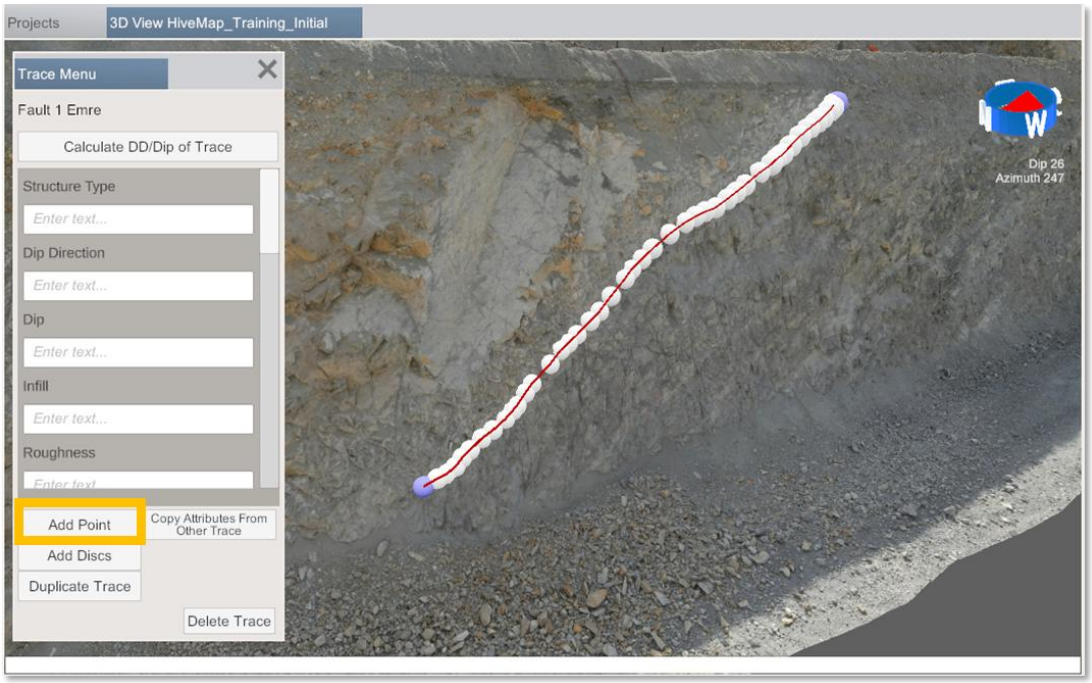


Step 2.7-8: The Trace Menu is displayed, and the nodes of the trace polyline/polygon becomes visible.



[Table of Contents]

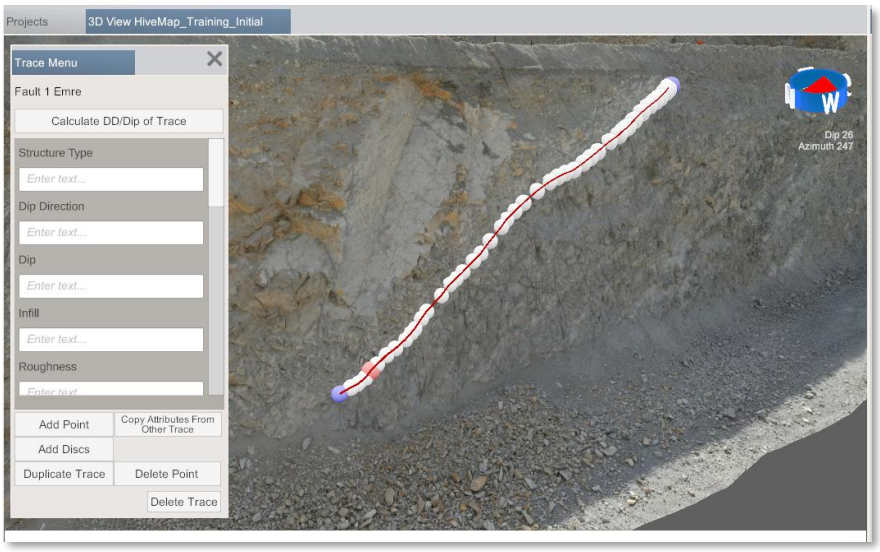
Step 2.7-9: Click the Add Point button and then click the trace line.



Step 2.7-10: A new node will be added where the line is clicked.

Table of Contents

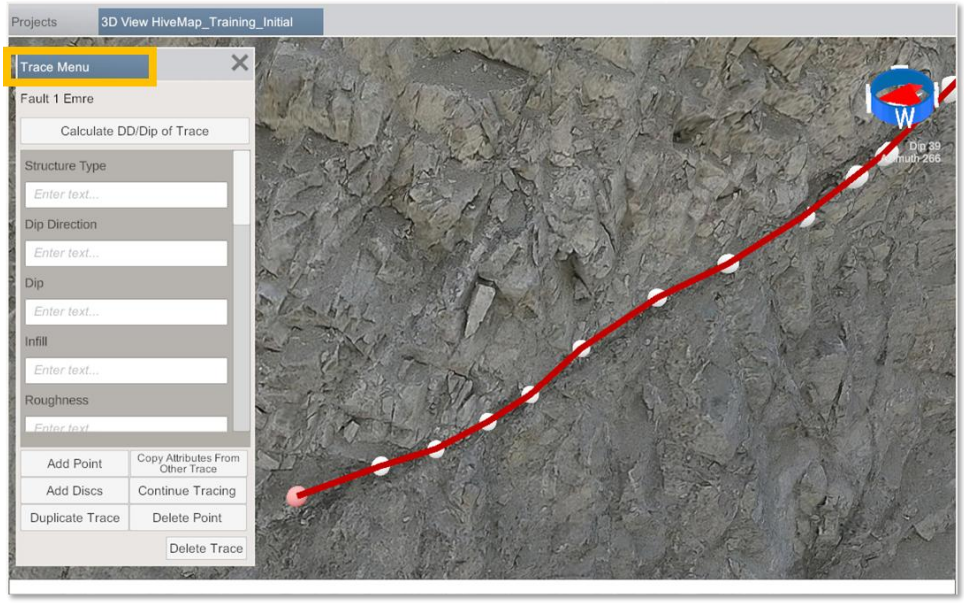
Step 2.7-11: Click one of the white nodes. The node will become red, and on the menu the Delete Point button will become visible. Click this button to delete the selected red node.



Step 2.7-12: Click the mesh and the red node will move to the clicked position on the mesh.

[Table of Contents]

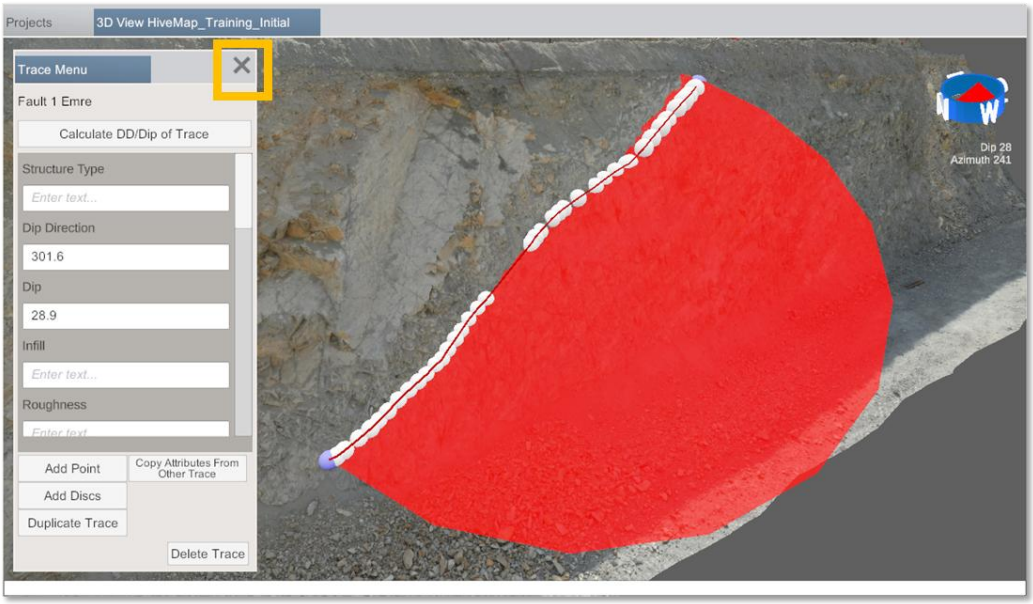
Step 2.7-13: Click the first or last node of the trace, which is coloured purple. The node will become red, and on the menu the Delete Point button and Continue Tracing button will become visible. Click Continue Tracing. This will close Trace Menu and hide the nodes and tracing will continue from the selected node.



Step 2.7-14: On the Trace Menu click the Calculate DD/Dip of Trace. A disc with the orientation of the structure will be created.

[Table of Contents]

Step 2.7-15: To save the changes click X on the top right of the Trace Menu. A menu will be displayed.



[Table of Contents](#)

2.8 Joint Mapping

Step 2.8-1 Go to the Mapping tab on the ribbon and click the Measure button on the Orientation Measurement group.
Step 2.8-2: Find a joint surface and start placing sampling points on it with your left mouse button. After placing a number of points, a disc will be created. A best fitting plane is calculated to find the orientation of the disc. Disc location and size is determined by the 2 furthest apart points.
Step 2.8-3: Number of points can be adjusted by using the Points slider. Increasing the number of sampling points will give more accurate orientation measurement.
Step 2.8-4: Continue mapping other joints.

- While placing sampling points you can press the Backspace key or click the Undo button to undo the last sampling point.
- If you have flat facets to map, you can adjust the point slider to minimum and create discs quickly by just placing 3 points for each of them.
- Alternatively, you can keep the Points slider at maximum and press the enter key to create discs by using variable number of points for each joint. If you have facets and traces with rough, wavey surfaces you need to increase the number of sampling points to get the correct average orientation of the joint.

Step 2.8-5: After finishing mapping click the Stop button.

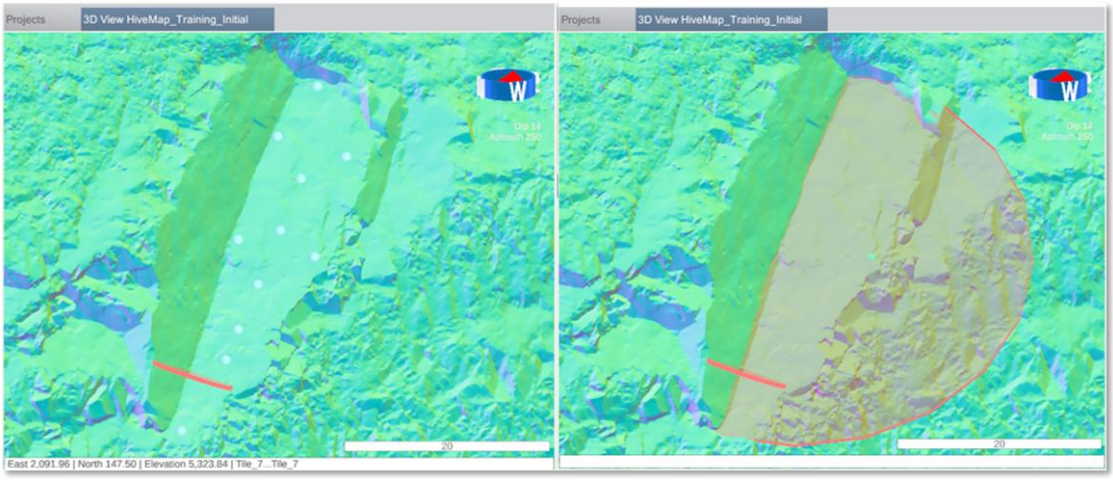
Disc data will be saved to OrientationData.csv file. If there is an unsaved data a * sign is displayed next to the file name in the explorer bar. Clicking the Save or Stop button saves data to the file and removes the * sign.

Step 2.8-1 Go to the Mapping tab on the ribbon and click the Measure button on the Orientation Measurement group.

Step 2.8-2: Find a joint surface and start placing sampling points on it with your left mouse button. After placing a number of points, a disc will be created. A best fitting plane is calculated to find the orientation of the disc. Disc location and size is determined by the 2 furthest apart points.

Step 2.8-3: Number of points can be adjusted by using the Points slider. Increasing the number of sampling points will give more accurate orientation measurement.

Table of Contents



Step 2.8-4: Continue mapping other joints.

- While placing sampling points you can press the Backspace key or click the Undo button to undo the last sampling point.
- If you have flat facets to map, you can adjust the point slider to minimum and create discs quickly by just placing 3 points for each of them.
- Alternatively, you can keep the Points slider at maximum and press the enter key to create discs by using variable number of points for each joint. If you have facets and traces with rough, wavey surfaces you need to increase the number of sampling points to get the correct average orientation of the joint.

Step 2.8-5: After finishing mapping click the Stop button.
Disc data will be saved to OrientationData.csv file. If there is an unsaved data a * sign is displayed next to the file name in the explorer bar. Clicking the Save or Stop button saves data to the file and removes the * sign.

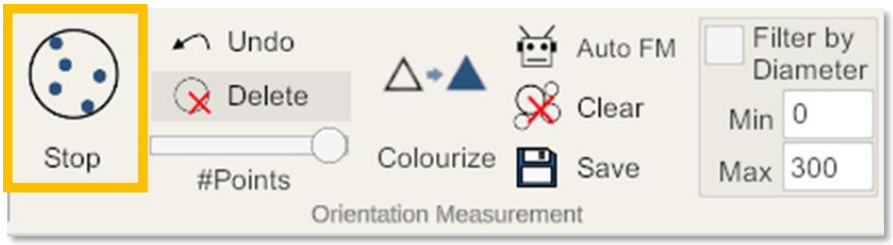


Table of Contents

2.9 Joint Set Analysis

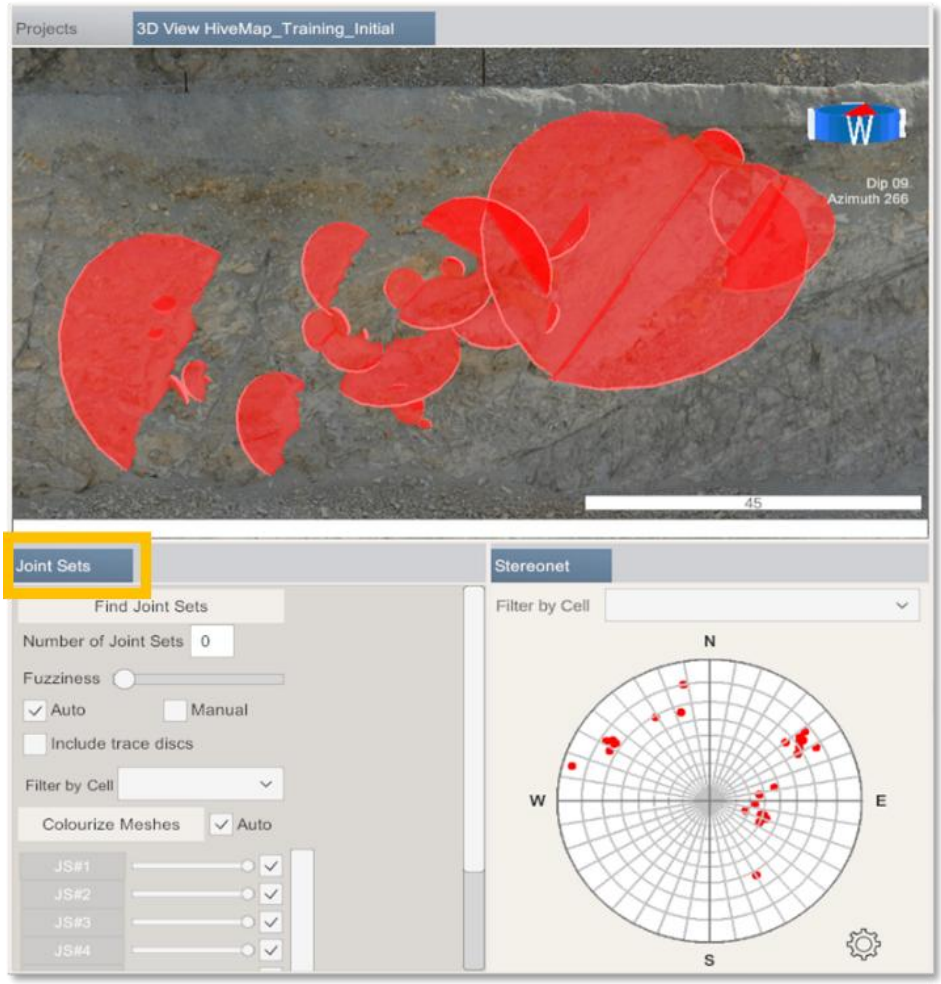
Step 2.9-1: Joint set analyses can be undertaken using the Joint Sets tab in the user interface.
Step 2.9-2: Adjust the Fuzziness slider to 0.70 and click the Find Joint Sets button. This separates joints which are away from the joint set centers and categorizes them as random joints.
Step 2.9-3: Increase the number of joint sets to 4 and run the algorithm again. This splits Joint Set 2 and some of the joints from this set become members of Joint Set 4.
Step 2.9-4: By following this process, adjusting the number of joint sets and fuzziness parameter, users can determine the joint sets with the help of this algorithm.
Step 2.9-5:To manually determine the joint sets, check the Manual box. Click the JS#1 button. This will open Joint Set 1 menu. Click the Select the Joint Set button on the Stereonet. Change the Dip Direction Deviation and Dip Deviation values on the form. Use your left mouse to change the location of the Joint Set 1 window on the Stereonet.
Step 2.9-6: Click X on the top right corner of the menu and click Save changes and close button. Click X on the top right corner of the menu and click Save changes and close button.
Step 2.9-7: Click the Assign Joints to Joint Sets button. All the joints inside Joint Set 1 window will be assigned to Joint Set 1.**:** Click the Assign Joints to Joint Sets button. All the joints inside Joint Set 1 window will be assigned to Joint Set 1.

Table of Contents

Step 2.9-1:

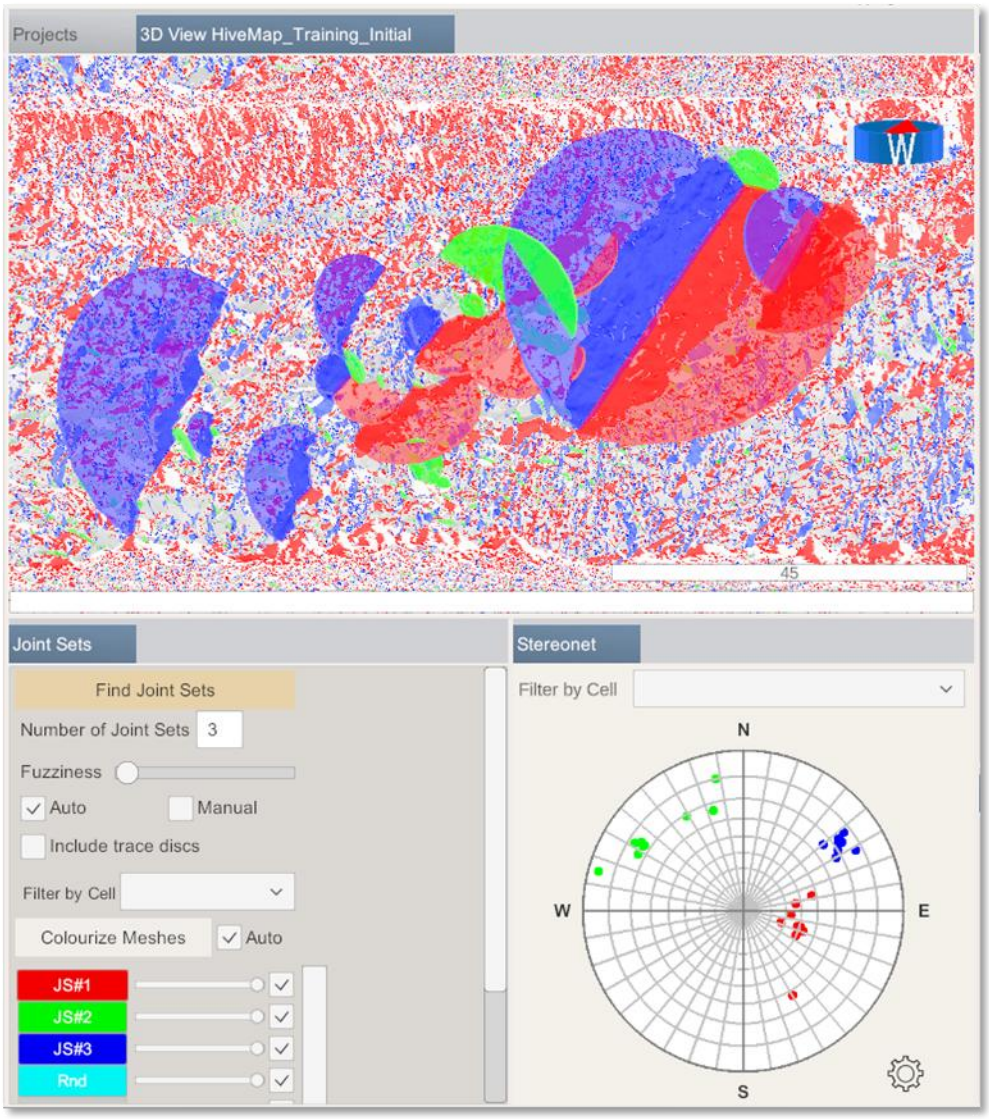
Joint set analyses can be undertaken using the Joint Sets tab in the user interface.

- The pole of each joint is displayed on the stereonet. From the rock face and the stereonet we can interpret that there are 3 or 4 joint sets.
- In the Joint Sets panel enter 3 to the Number of Joint Sets box. Then click the Find Joint Sets button.
- The algorithm separates joints into 3 sets and colourizes each joint pole on the stereonet and the disc on the 3D View in their joint set colour. Also, the triangles on the mesh are colourized if they have the same orientation as the joint sets.



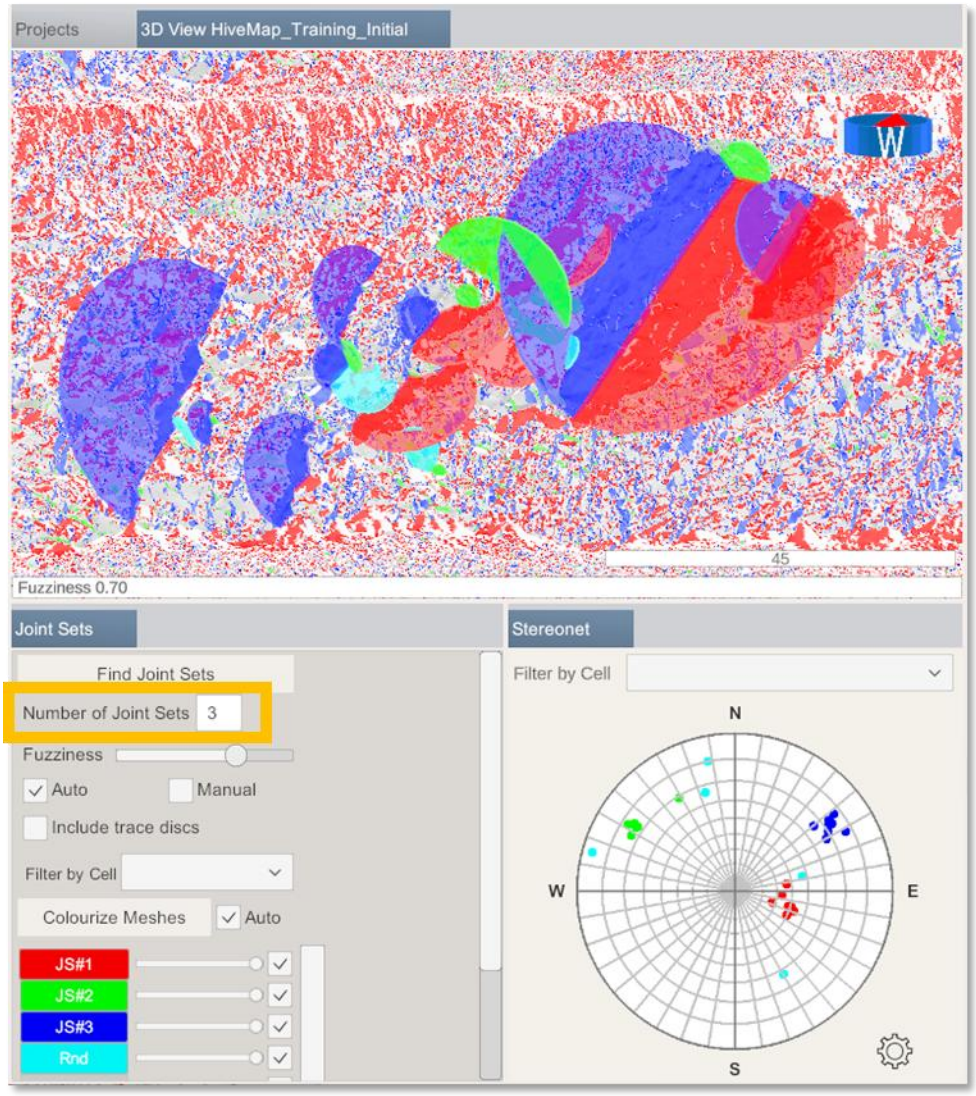
[Table of Contents]

Step 2.9-2: Adjust the Fuzziness slider to 0.70 and click the Find Joint Sets button. This separates joints which are away from the joint set centers and categorizes them as random joints.



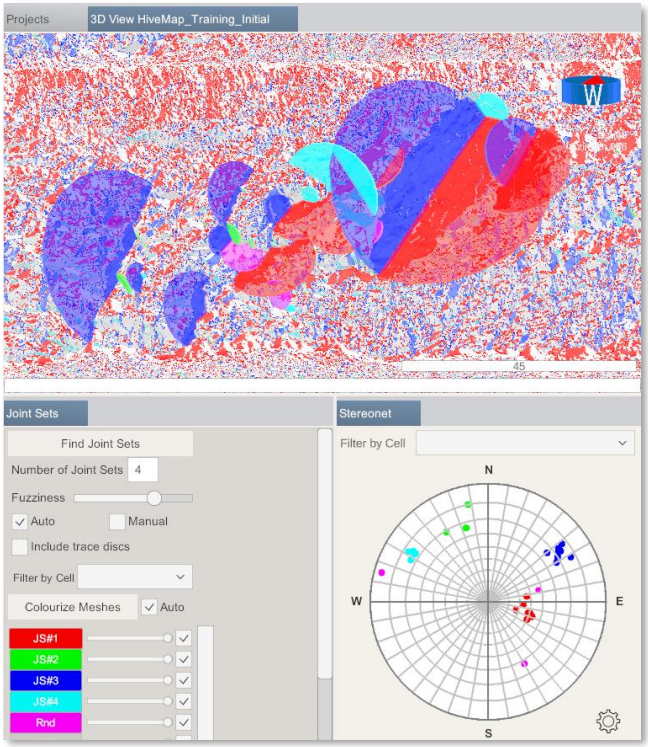
[Table of Contents](#)

Step 2.9-3: Increase the number of joint sets to 4 and run the algorithm again. This splits Joint Set 2 and some of the joints from this set become members of Joint Set 4.

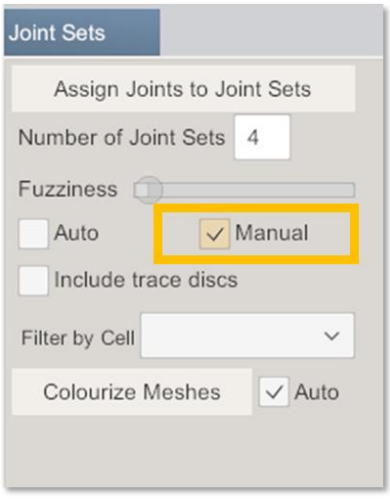


[Table of Contents]

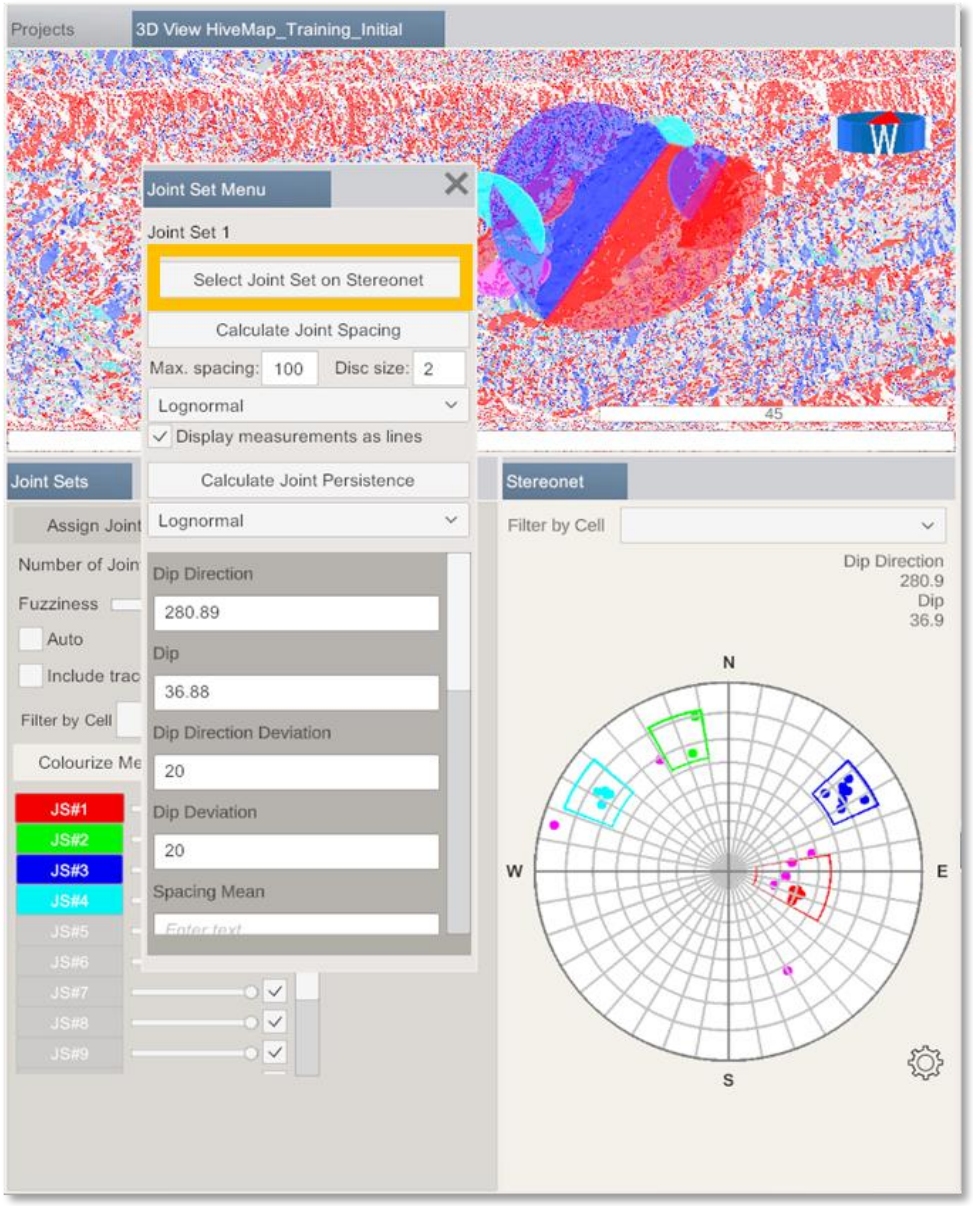
Step 2.9-4: By following this process, adjusting the number of joint sets and fuzziness parameter, users can determine the joint sets with the help of this algorithm.



Step 2.9-5: To manually determine the joint sets, check the Manual box. Click the JS#1 button. This will open Joint Set 1 menu. Click the Select the Joint Set on Stereonet. Change the Dip Direction Deviation and Dip Deviation values on the form. Use your left mouse to change the location of the Joint Set 1 window on the Stereonet.

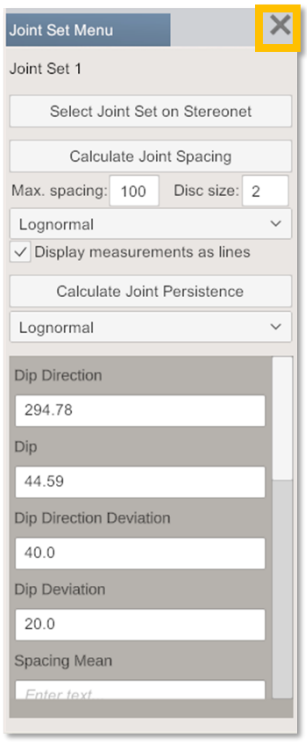


[Table of Contents]

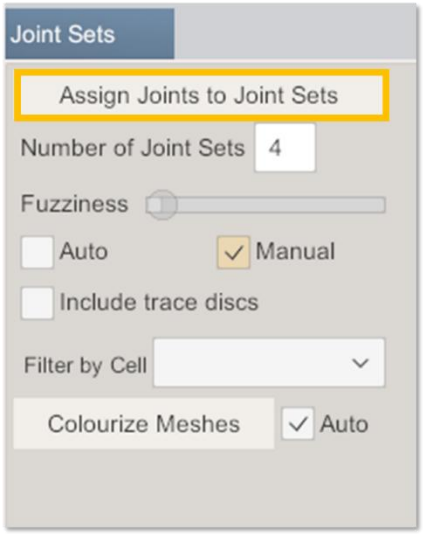


[Table of Contents](#)

Step 2.9-6: Click X on the top right corner of the menu and click Save changes and close button.

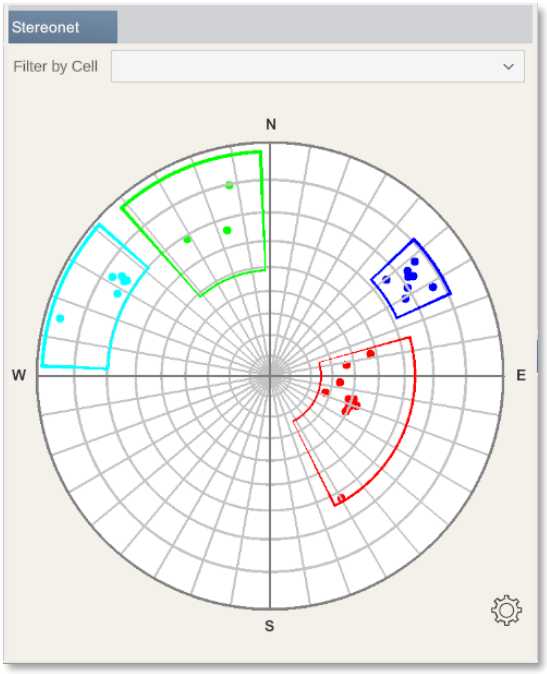


Step 2.9-7: Click the Assign Joints to Joint Sets button. All the joints inside Joint Set 1 window will be assigned to Joint Set 1.



[Table of Contents]

Step 2.9-8: Repeat this process for the other joint sets.

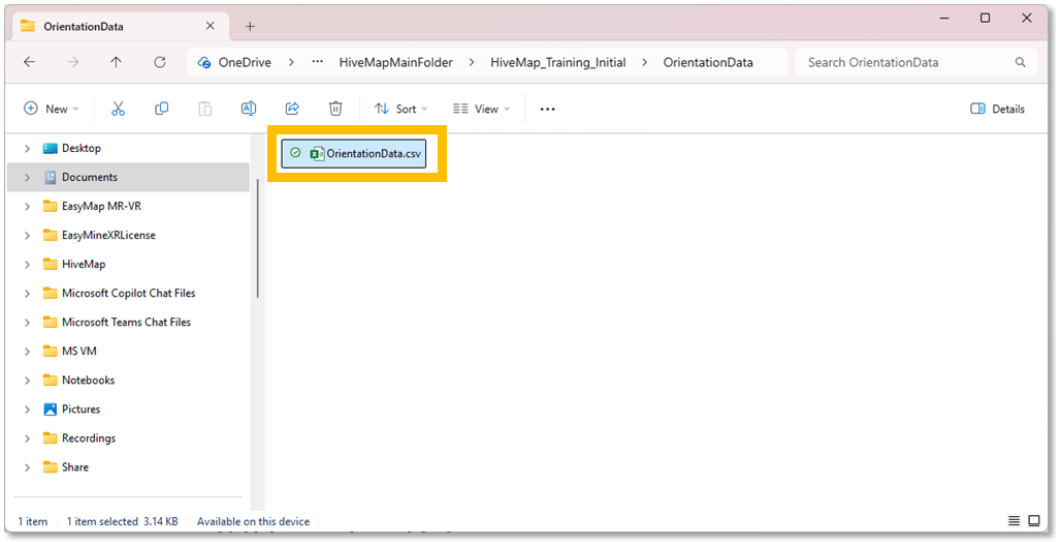


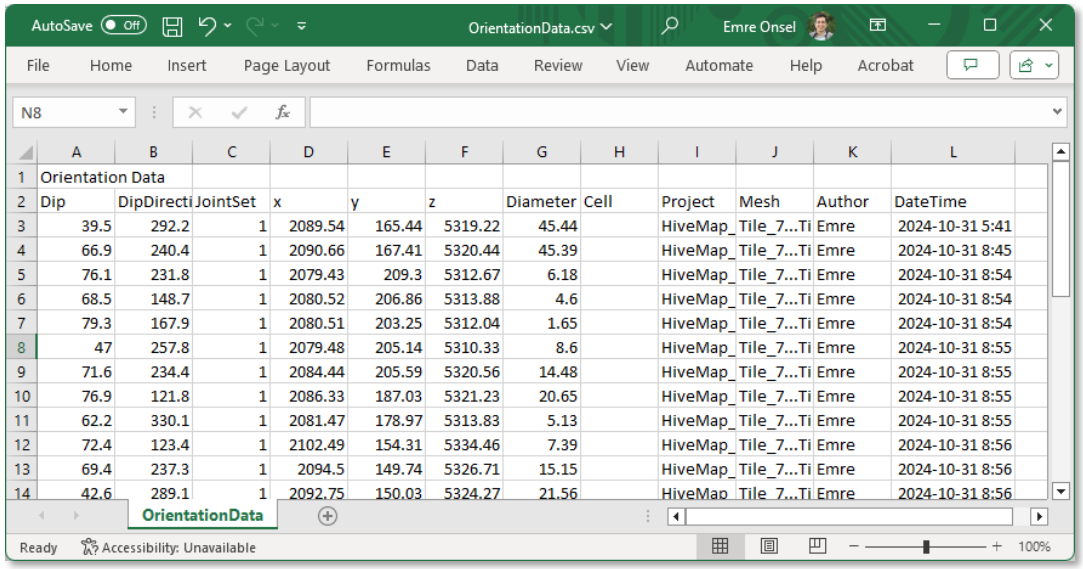
2.10 Exporting Data

2.10.1 Folder Structure

2.10.2 Joint Mapping Data

Run Windows File Explorer and go to the main HiveMap folder. Open the project folder “HiveMap_Training_Initial”. Then go to OrientationData folder which has OrientationData.csv file.
This file contains all the data which is captured with the Orientation measurement tool.



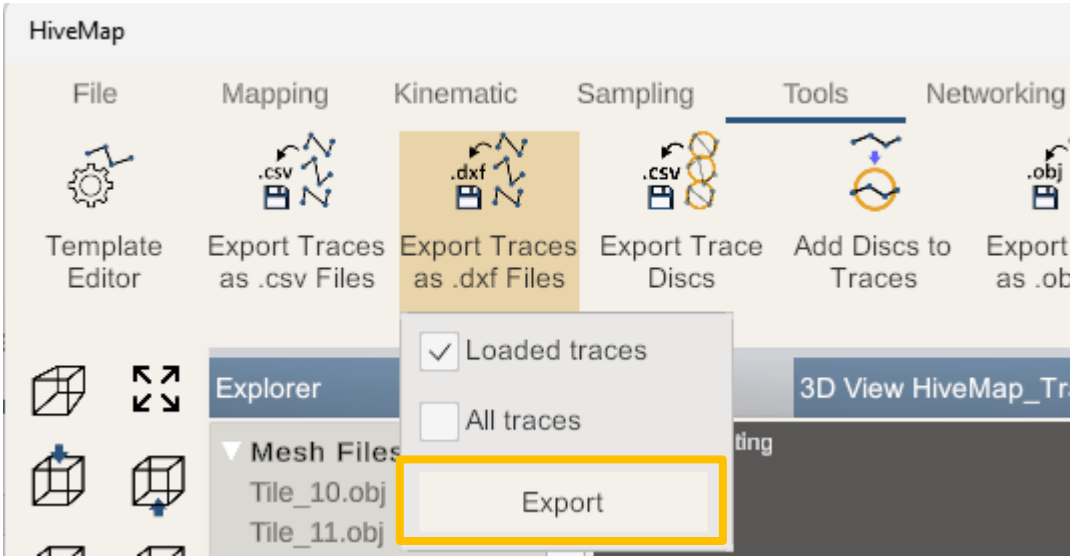


2.10.3 Tracing Data

Step 2.10-1: Go to the Tools menu. Click Export Traces as .dxf Files button. Click Export button. This will export polyline/polygon of the traces to .dxf files.
Step 2.10-2: Click the Export Trace Discs button.

Table of Contents

Step 2.10-1: Go to the Tools menu. Click Export Traces as .dxf Files button. Click Export button. This will export polyline/polygon of the traces to .dxf files.



These .dxf files can be found in the ExportedTraceData folder of the project folder.

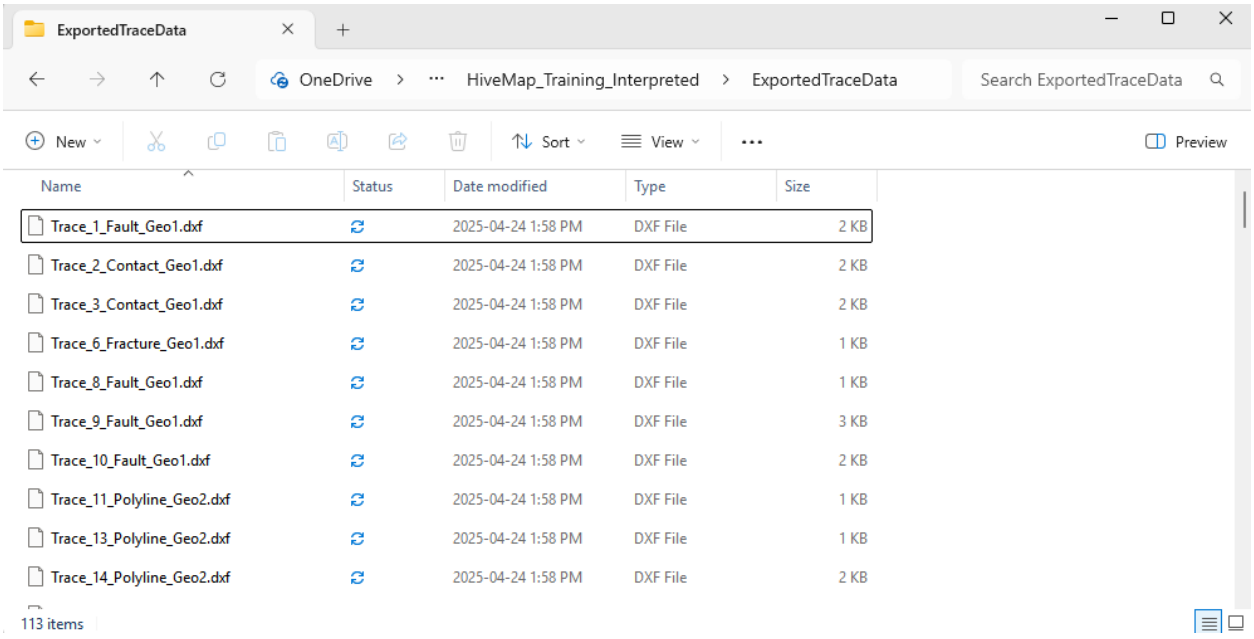
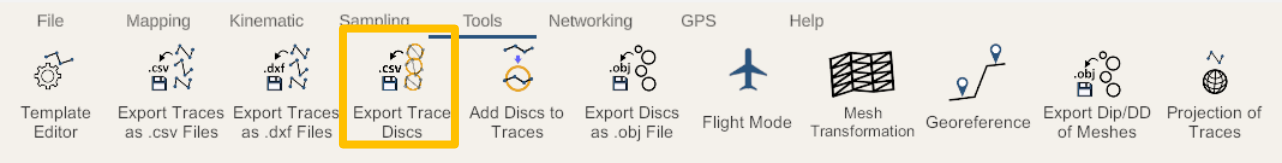
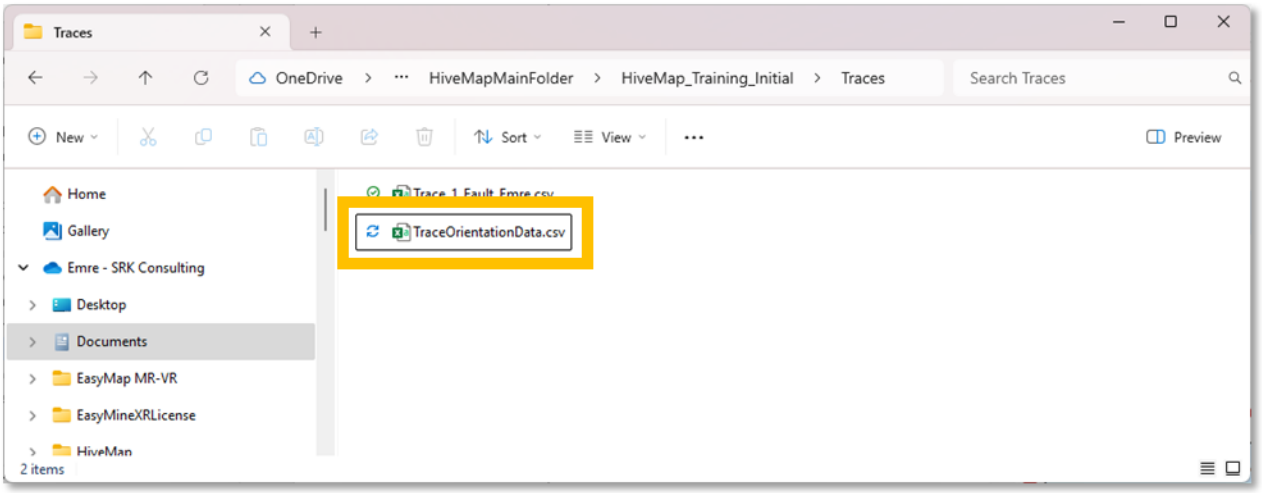


Table of Contents

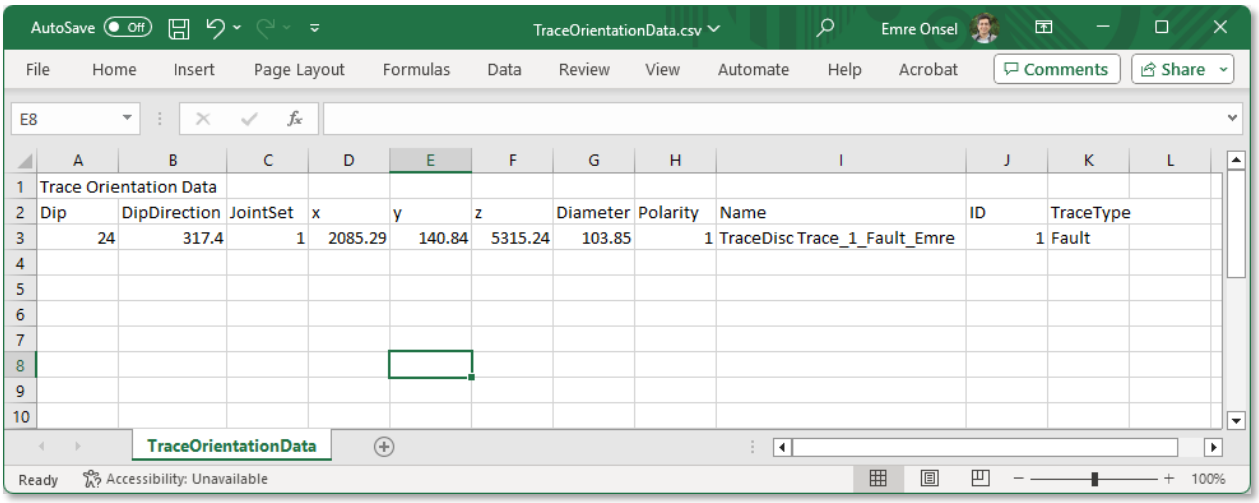
Step 2.10-2: Click the Export Trace Discs button.



This will create the TraceOrientationData.csv in the ExportedTraceData folder of the project.



This file has all the discs from the traces.



3 User Guide

3.1 Projects

Step 3.1-1: Projects are managed in the Projects panel. If the projects panel is not visible click the File button on the ribbon or click title of the Projects panel.: Projects are managed in the

Table of Contents

Projects panel. If the projects panel is not visible click the File button on the ribbon or click title of the Projects panel.

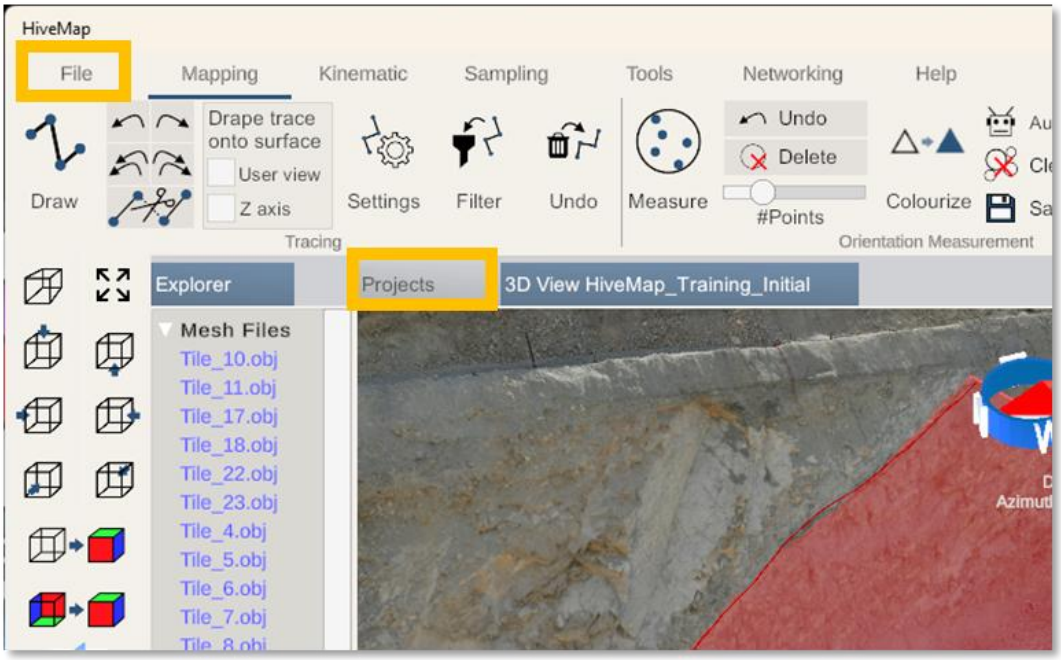
Step 3.1-2: The last 4 projects that are opened in HiveMap are listed on the top of the panel as buttons. Left clicking these buttons will load that project. The last 4 projects that are opened in HiveMap are listed on the top of the panel as buttons. Left clicking these buttons will load that project.

project. All projects are listed in the Projects list. In this list are the subfolders which are in the HiveMap main folder (Section 2.2). Left clicking these buttons will load that project.

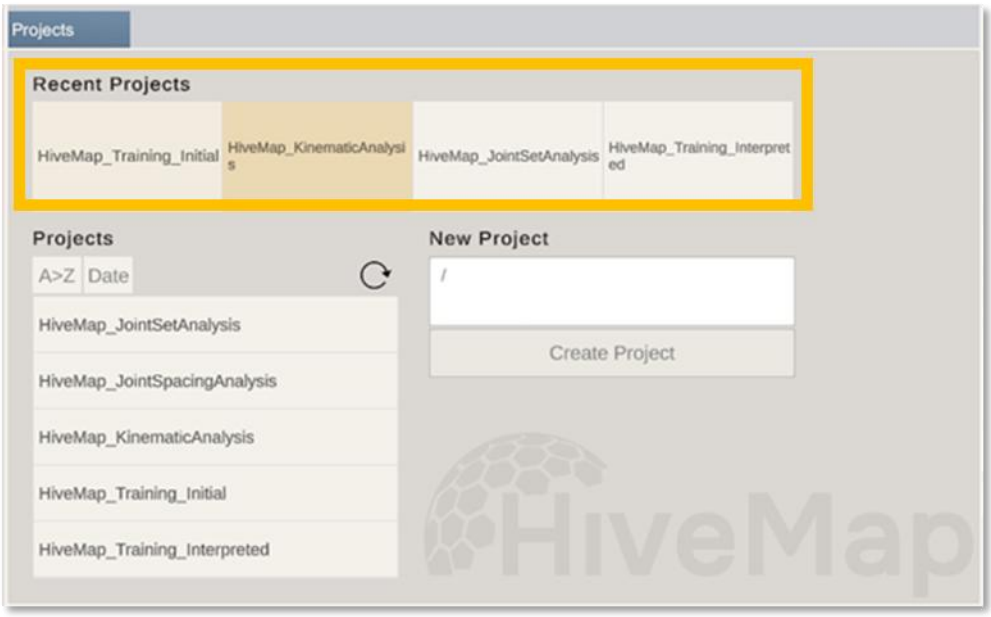
list. By using A>Z or Date buttons the list can be sorted in alphabetical order or according to project's last edit date. If there are new subfolders in the main folder, click the circular arrow button to update the list.

Project. Create a new project by entering the New Project name and selecting Create Project. This will prompt the display view to the 3D View.

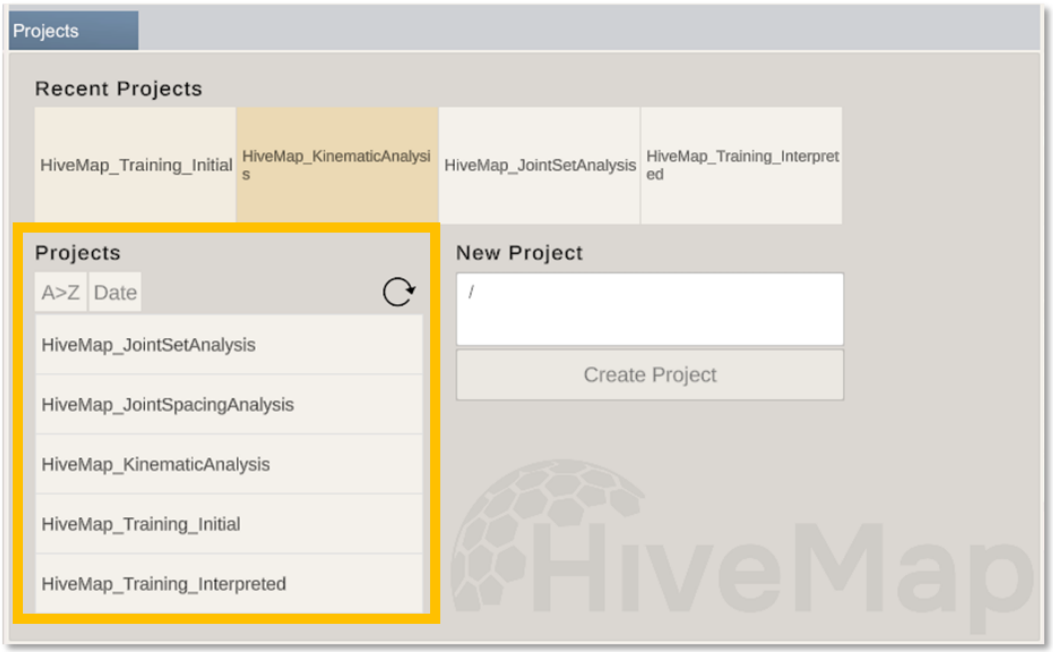
Step 3.1-1: Projects are managed in the Projects panel. If the projects panel is not visible click the File button on the ribbon or click title of the Projects panel.



Step 3.1-2: The last 4 projects that are opened in HiveMap are listed on the top of the panel as buttons. Left clicking these buttons will load that project.



Step 3.1-3: All projects are listed in the Projects list. In this list are the subfolders which are in the HiveMap main folder (Section 2.2). Left clicking these buttons will load that project.

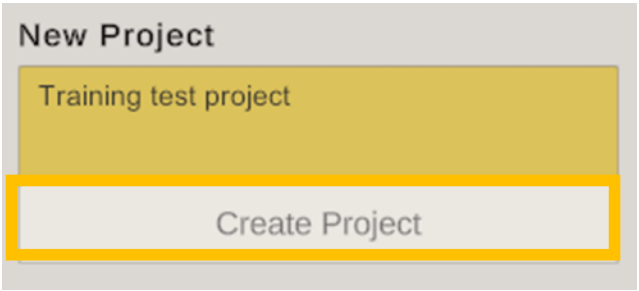


Step 3.1-4: By using A>Z or Date buttons the list can be sorted in alphabetical order or according to project's last edit date. If there are new subfolders in the main folder, click the circular arrow button to update the list.

Table of Contents

Step 3.1-5: Create a new project by entering the New Project name and selecting Create Project.

This will prompt the display view to the 3D View.

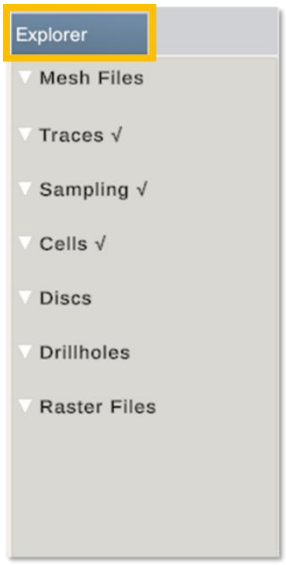


3.2 Project Explorer and Data

Error! Reference source not found.The explorer panel displays all the imported and created data for the projects, with each data type organized under its own branch. These branches include Mesh files (link to section), Traces (link to Mapping section), Sampling (link to Sampling section), Cells (link), Discs (link), Drillholes (link), and Raster files (link).
Step 3.2-1: Select All selects all the items in the Explorer. Select All selects all the items in the Explorer.
Step 3.2-2: Left or right clicking a branch selects the items that belongs to that branch. By using the Ctrl and Shift Keys with left click, multiple items can be selected.

Step 3.2-1: The explorer panel displays all the imported and created data for the projects, with each data type organized under its own branch. These branches include: Mesh files , Traces , Sampling , Cells , Discs, Drillholes, and Raster files.

Table of Contents

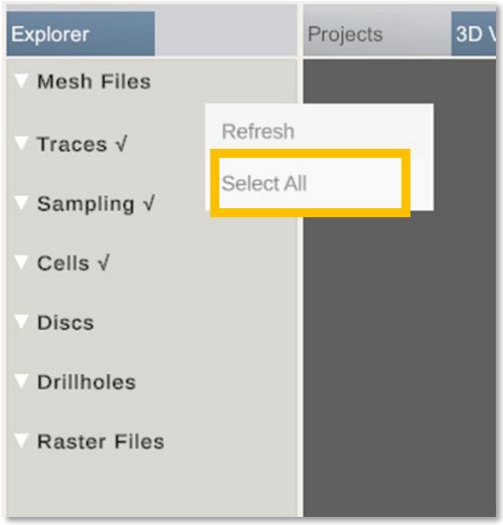


Right clicking the Explorer panel will display the right click menu. This menu will change depending on what kind of item is clicked.

In an empty area, the right click menu will display Refresh and Select All. Refresh recreates the project explorer list.

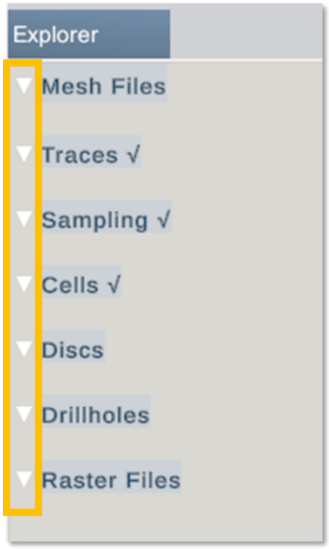
If there is a change in the project folder such as removed or added files, that change will be seen in the Explorer after Refresh.

Step 3.2-1: Select All selects all the items in the Explorer.



Step 3.2-2: Left or right clicking a branch selects the items that belongs to that branch.

By using the Ctrl and Shift Keys with left click, multiple items can be selected.



3.2.1 Training Data Files

Users can access HiveMap training data files on the website Resources page:
hivemap3d.com/resources

The HiveMap Training Data.zip file folder contains 3 zipped sub-folders as follows:

- HiveMap_JointSetAnalysis
- HiveMap_Training_Initial
- HiveMap_Training_Interpreted

Error! Reference source not found.
Error! Reference source not found. Extract the files for each zipped sub folder.

These files will be used during HiveMap system training when a project is created, and to demo other system functions that are being reviewed during HiveMap training.

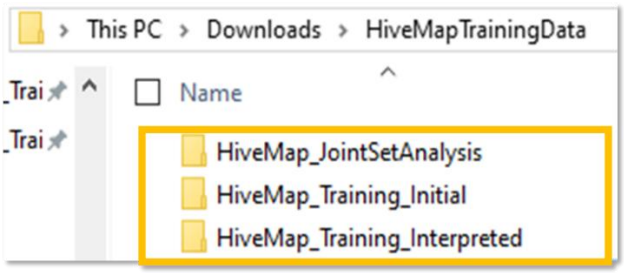


Table of Contents

3.2.2 Mesh Files

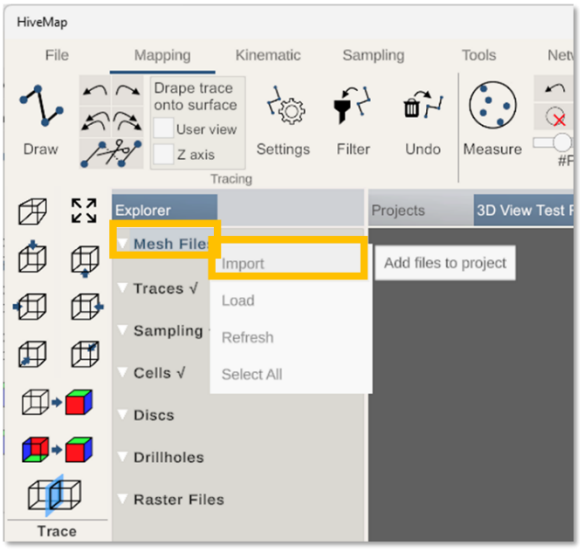
HiveMap works with textured mesh files which are in .obj file format. The file formats are as follows:

- .obj files are created with 2 more files which are .mtl and .jpg files
- .obj files have the geometry information such as triangles, vertices and vertex normals
- .mtl files are text files referenced by the .obj file, which have the list of texture files
- .jpg files are texture files. There can be more than 1 file. Textures are limited to 16384 x 16384 pixels.

Step 3.2-6: After creating a new project, to import mesh files, from within the Explorer panel right click over the Mesh Files branch and select Import.
Step 3.2-7: The File Explorer window will pop up. Find and select the mesh files (.obj) and side files (.mtl, .jpg). To multiselect more than one file use the Ctrl or Shift keys.
Loaded meshes will appear in blue in the Explorer panel. When you right-click on a mesh file, the context menu will show options to Zoom, Hide, and Unhide.

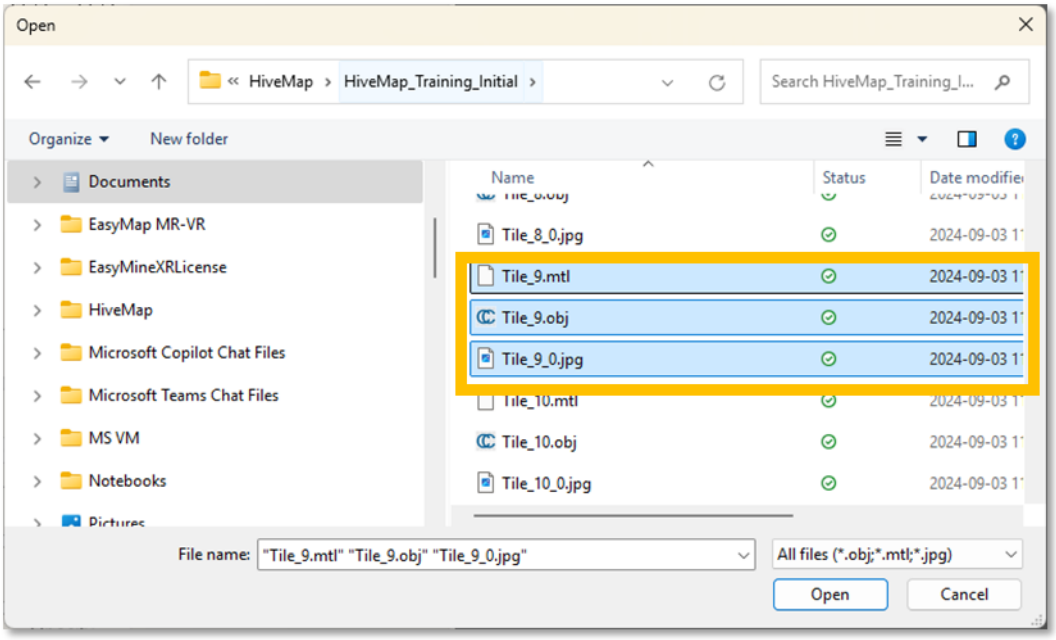
Step 8: Zoom will zoom to the selected mesh.
Step 3.2-9: Hide and Unhide will hide or unhide the selected meshes. Hidden meshes will be displayed in a grayed-out appearance.
Step 3.2-: Left click on a mesh to view its properties in the Properties panel. From there, you can hide or unhide the mesh, adjust its brightness and opacity, and change its color. Please note that these changes will be reset after reloading the meshes.
To change the mesh colour, go to the View menu and select either "Texture" or "Shade" from the attribute map list.
Step 3.2-31: Left click the mesh.
On the properties tab click Change Mesh Colour button. A menu with colour options will pop up. Click the colour that you want.

Step 3.2-6: After creating a new project, to import mesh files, from within the Explorer panel right click over the Mesh Files branch and select Import.

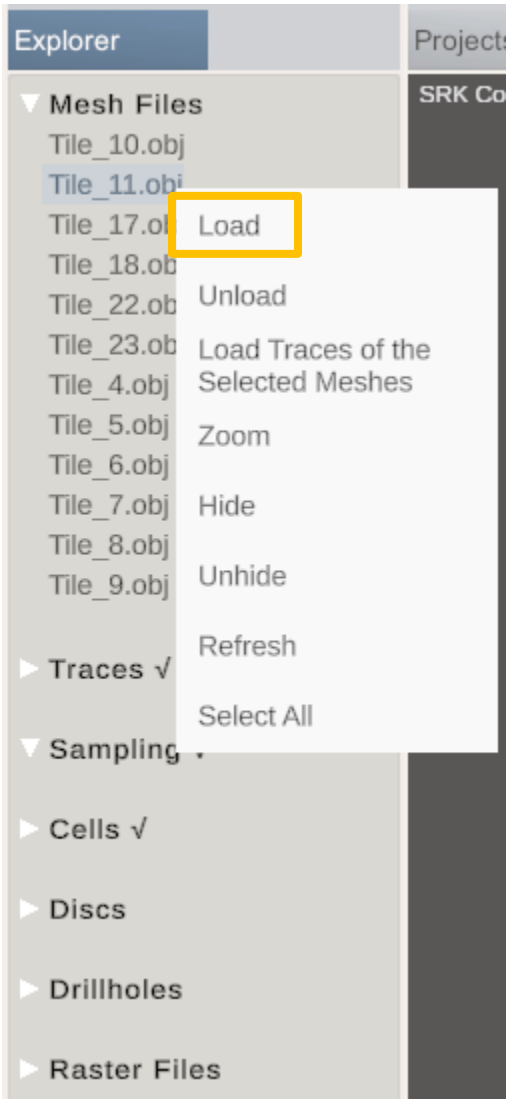


Step 3.2-7: The File Explorer window will pop up. Find and select the mesh files (.obj) and side files (.mtl, .jpg). To multiselect more than one file use the Ctrl or Shift keys.
The imported files will be listed in Explorer. Right click the mesh files and select Load to display the meshes in 3D View.

[Table of Contents]


Open
HiveMap
HiveMap_Training_Initial
Search HiveMap_Training_I...
Organize
New folder
Documents
EasyMap MR-VR
EasyMineXRLicense
HiveMap
Microsoft Copilot Chat Files
Microsoft Teams Chat Files
MS VM
Notebooks
Pictures
Name
Status
Date modifie
Tile_8_0.jpg
Tile_9.mtl
Tile_9.obj
Tile_9_0.jpg
Tile_10.mtl
Tile_10.obj
Tile_10_0.jpg
2024-09-03 1
File name: "Tile_9.mtl" "Tile_9.obj" "Tile_9_0.jpg"
All files (*.obj;*.mtl;*.jpg)
Open
Cancel

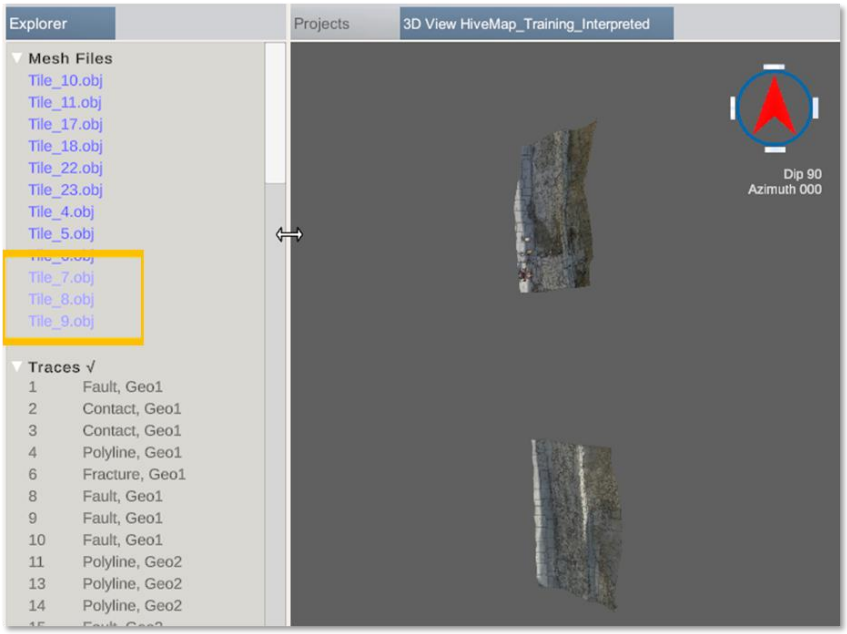
Loaded meshes will appear in blue in the Explorer panel. When you right-click on a mesh file, the context menu will show options to Zoom, Hide, and Unhide.



[Table of Contents](#)

Step 3.2-8: Zoom will zoom to the selected mesh.

Step 3.2-9: Hide and Unhide will hide or unhide the selected meshes. Hidden meshes will be displayed in a grayed-out appearance.



Step 3.2-10: Left click on a mesh to view its properties in the Properties panel. From there, you can hide or unhide the mesh, adjust its brightness and opacity, and change its color. Please note that these changes will be reset after reloading the meshes.
To change the mesh colour, go to the View menu and select either "Texture" or "Shade" from the attribute map list.

[Table of Contents]

Step 3.2-31: Left click the mesh.
On the properties tab click Change Mesh Colour button. A menu with colour options will pop up. Click the colour that you want.

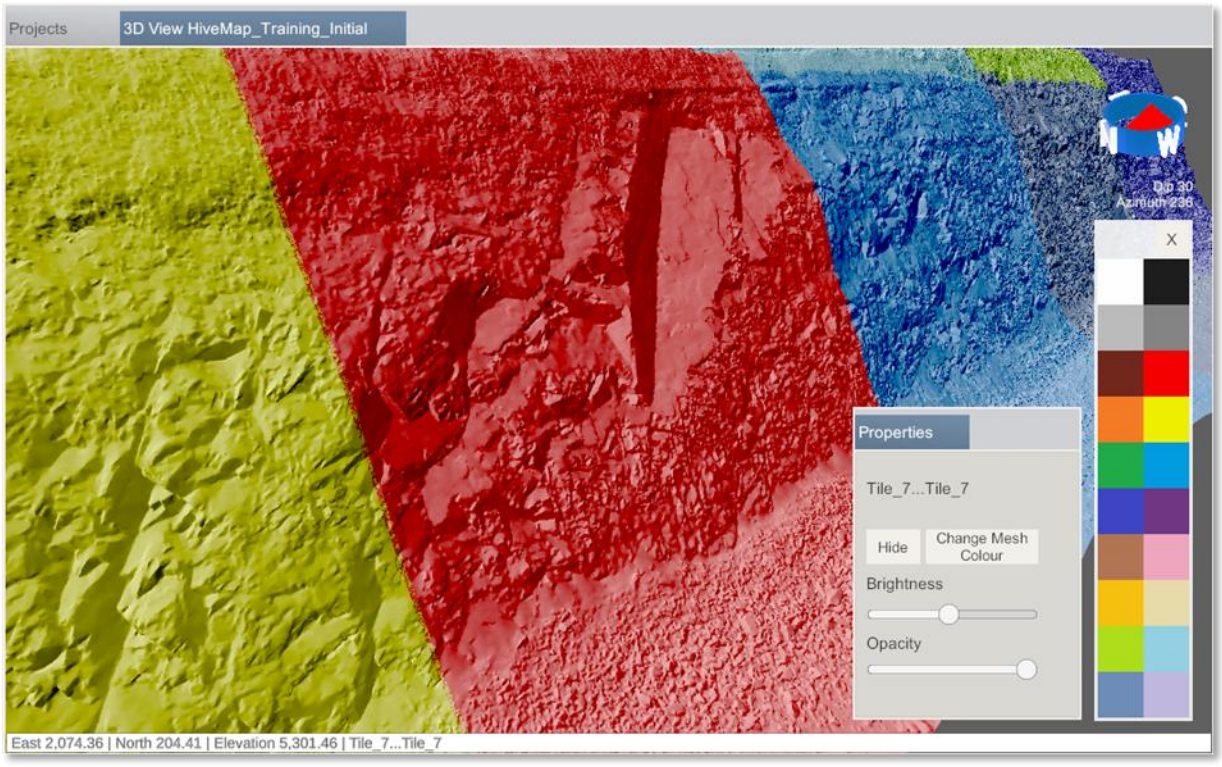


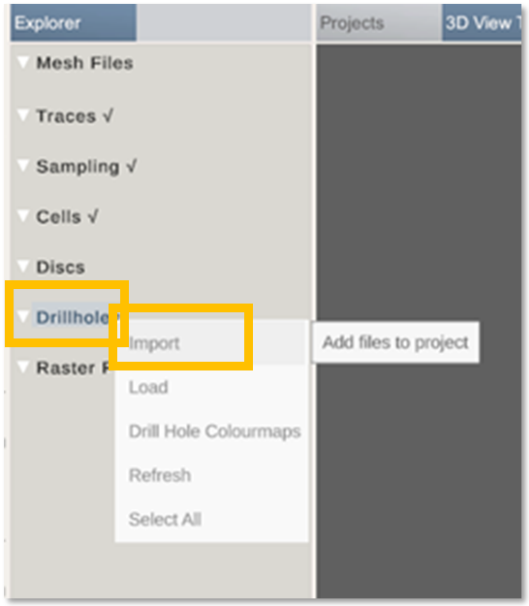
Table of Contents

3.2.3 Drill Holes

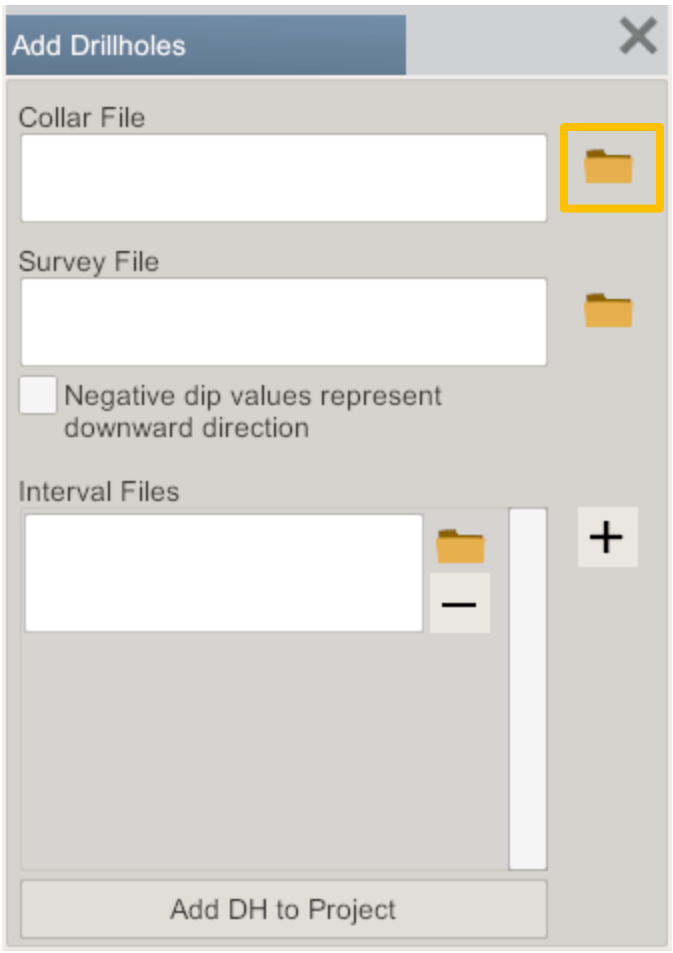
Step 3.2-42: To import drill holes data, go to Explorer panel and right click Drillholes branch and select Import.To import drill holes data, go to Explorer panel and right click on the Drillholes branch and select Import.
Step 3.2-53: The following Add Drill Holes pop-up menu will appear. Click the folder icon next to Collar File.
Step 3.2-64: Select the collar file, which is in .csv file format.
Step 3.2-65: The first 5 rows of the file are listed. Select the correct matching columns for the Hole ID, X, Y, Z and depth parameters of the collar data.
Step : Repeat the same process for the survey file. Select the correct matching columns for the Hole ID, Depth, Dip and Azimuth parameters of survey data.
You can have more than one interval file. Click the plus and minus icons to adjust the number of interval files.
Step 3.2-97: Click the folder icon next to Interval Files to select the interval file. Select the correct matching columns for the Hole ID, From and To parameters of the interval data. Select “Value” for the columns that you want to import. If “Ignore” is selected for a column that column will not be imported.
Step 3.2-108: Once the collar, survey and interval files are added, click the Add DH to Project button.
Step 3.2-119: Drillholes will appear in the Explorer panel under the Drillholes branch. Right click the Drillholes and select the Drill Hole Colourmaps.
Step 3.2-20: If the data type is a category, each unique value will be identified and randomly assigned a colour.
Step 3.2-121: To manually create the drillhole colour map, click the Add Colour map item, which will add a new row. Clicking the minus icon in the row will remove the row. If the data type is numeric, type numbers into the Property box. If the data type is category, type the values into the Property box. Then click the Colour box and select the colour.
Step 3.2-132: After creating the colourmap, automatically or manually click the Save Colourmap button.
Step 3.2-143: To load the drillholes select the drillholes in Explorer, right click and select Load.

Step 3.2-42: To import drill holes data, go to Explorer panel and right click Drillholes branch and select Import.

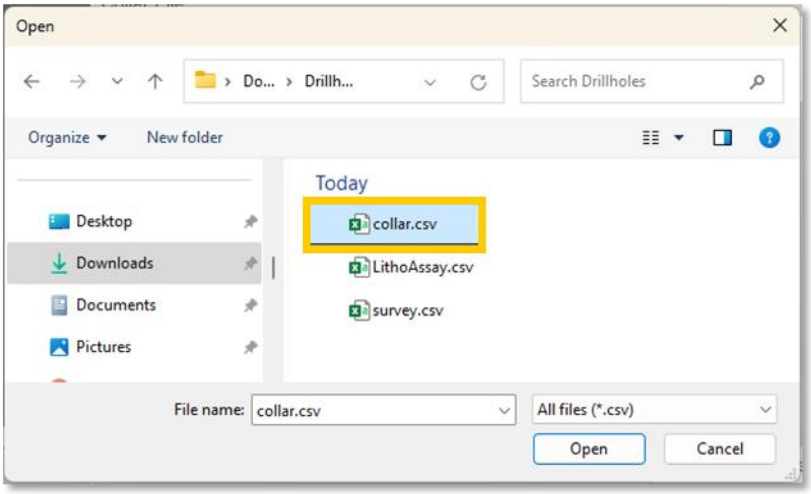
Table of Contents



Step 3.2-53: The following Add Drill Holes pop-up menu will appear. Click the folder icon next to Collar File.

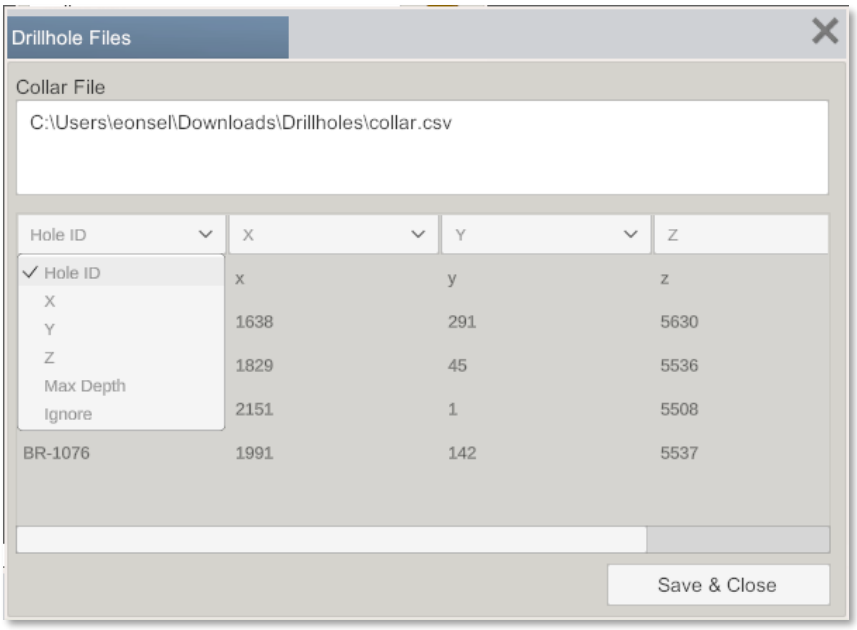


Step 3.2-64: Select the collar file, which is in .csv file format.



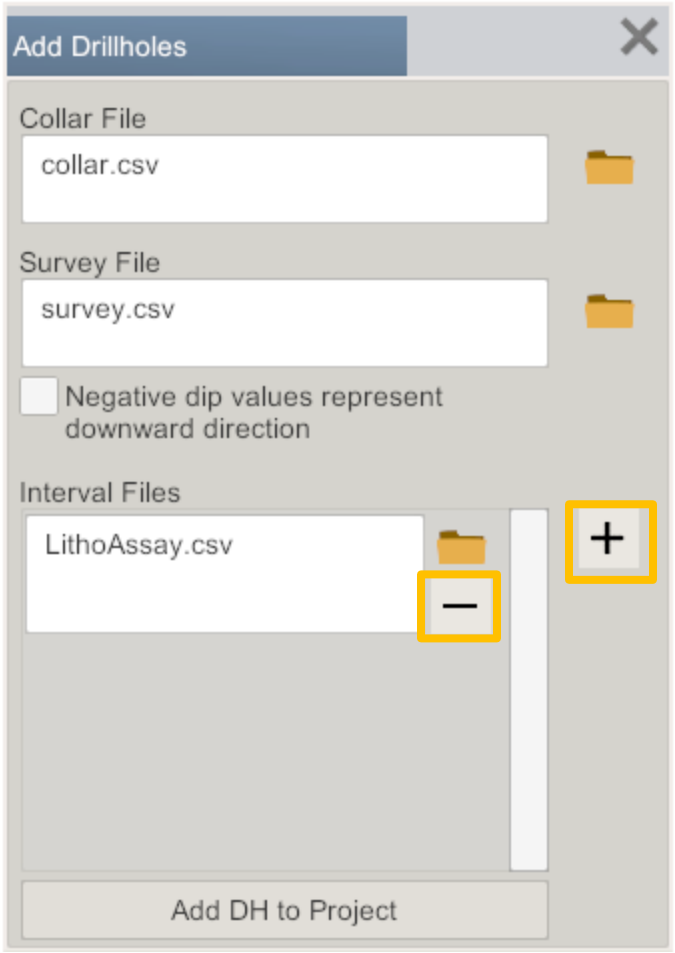
Step 3.2-75: The first 5 rows of the file are listed. Select the correct matching columns for the Hole ID, X, Y, Z and depth parameters of the collar data.

Table of Contents



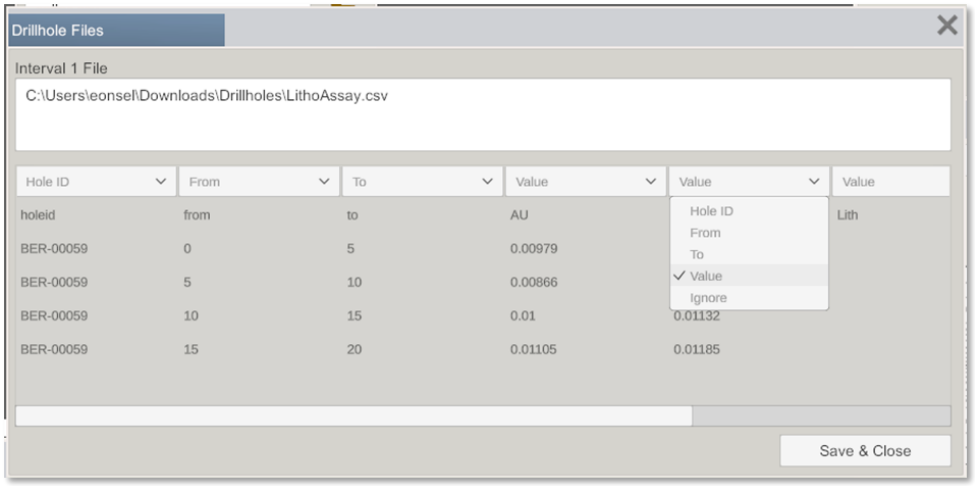
Step 3.2-86: Repeat the same process for the survey file. Select the correct matching columns for the Hole ID, Depth, Dip and Azimuth parameters of survey data. Check the “Negative dip values represent downward direction” checkbox if this applies to your data.

You can have more than one interval file. Click the plus and minus icons to adjust the number of interval files.



Step 3.2-97: Click the folder icon next to Interval Files to select the interval file. Select the correct matching columns for the Hole ID, From and to parameters of the interval data. Select “Value” for the columns that you want to import. If “Ignore” is selected for a column that column will not be imported.

Table of Contents



Step 3.2-108: Once the collar, survey and interval files are added, click the Add DH to Project button.

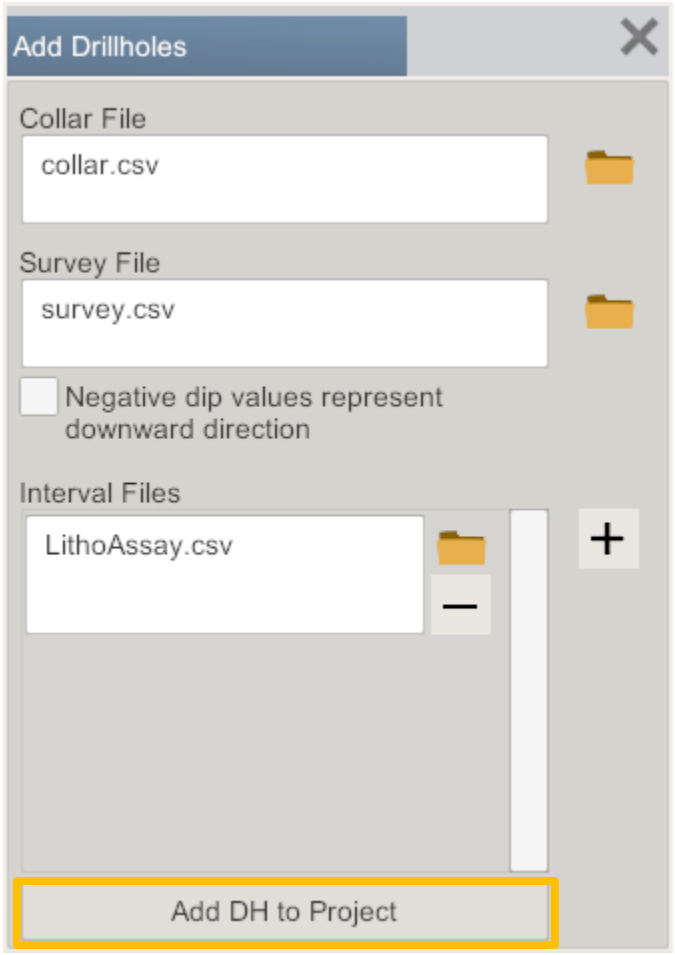
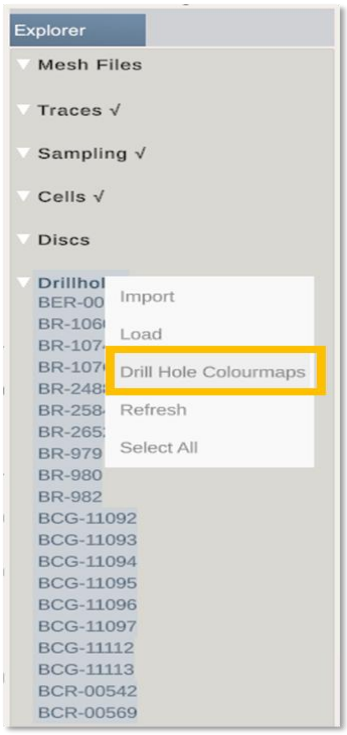


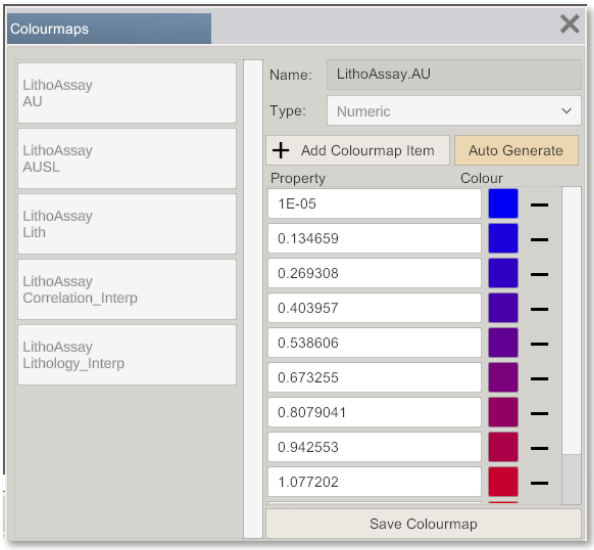
Table of Contents

Step 3.2-119: Drillholes will appear in the Explorer panel under the Drillholes branch. Right click the Drillholes and select the Drill Hole Colourmaps.

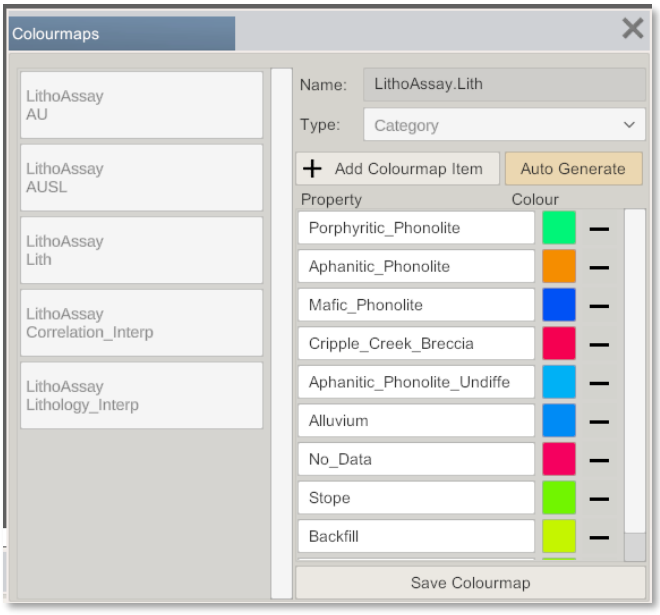


Select the interval property and select the type of data as Numeric or Category. Click Auto Generate. If the data type is numeric, the minimum and maximum values will be found, and the data will be split into 10 equally sized groups; coloured from blue to red.

Step 3.2-20: If the data type is a category, each unique value will be identified and randomly assigned a colour.

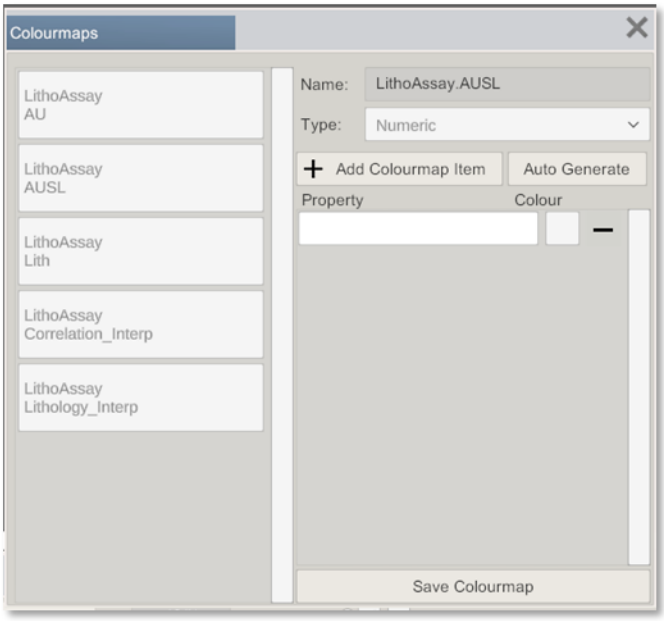


Step 3.2-121: To manually create the drillhole colour map, click the Add Colourmap item, which will add a new row. Clicking the minus icon in the row will remove the row. If the data type is numeric, type numbers into the Property box. If the data type is category, type the values into the Property box. Then click the Colour box and select the colour.

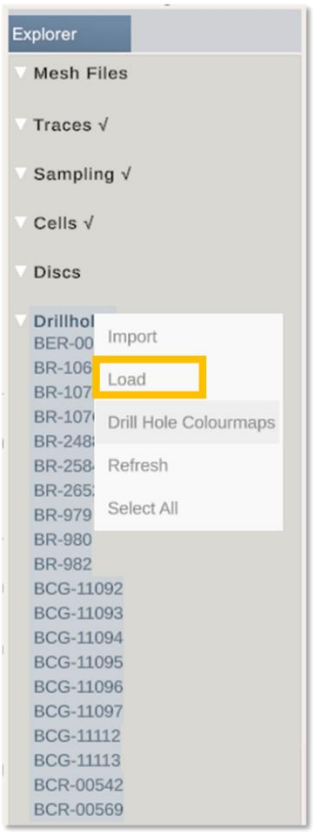


[Table of Contents]

Step 3.2-132: After creating the colourmap, automatically or manually click the Save Colourmap button.



Step 3.2-143: To load the drillholes select the drillholes in Explorer, right click and select Load.



[Table of Contents]

3.2.4 Raster Files

Step 3.2-154: GeoTiff files can be imported to HiveMap. They are limited to 16384 x 16384 pixels. To import raster files, in the Explorer panel right click the Raster Files branch and select Import.
Step 3.2-165: Right click the imported raster files and click load to display them on 3D View.
Step 3.2-176: The raster file will appear as a georeferenced image in the 3D View. It will be at 0 altitude.
Step 3.2-187: Raster files can be draped to meshes. Right click the raster file and select Drape to Mesh.
Step 3.2-198: On the Explorer panel everything becomes grayed except the mesh files. Click the mesh that you want to drape the raster file to.
Step 3.2-9: The name of the mesh file will be displayed next to Raster Files. If the mesh is not loaded. Load the mesh and raster file. Raster files will be displayed on the mesh.

Step 3.2-154: GeoTiff files can be imported to HiveMap. They are limited to 16384 x 16384 pixels.To import raster files, in the Explorer panel right click the Raster Files branch and select Import.

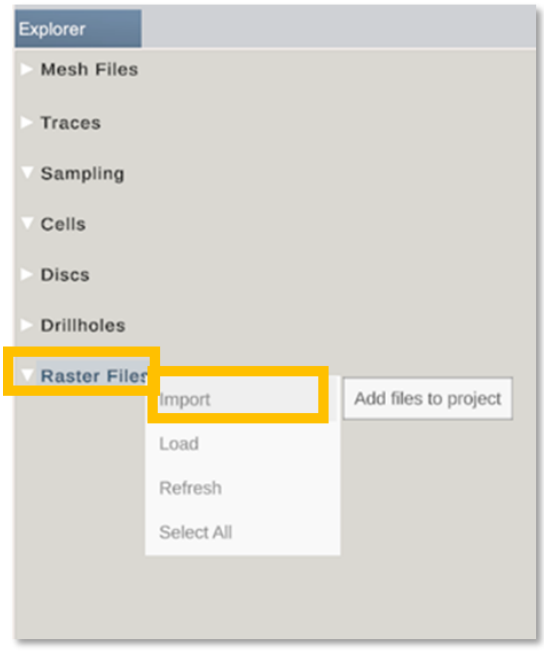
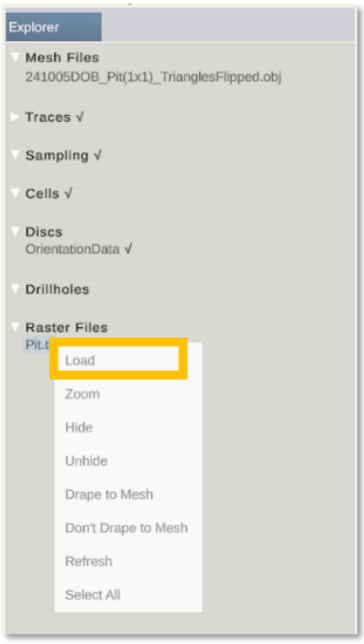


Table of Contents

Step 3.2-16: Right click the imported raster files and click load to display them on 3D View.



Step 3.2-17: The raster file will appear as a georeferenced image in the 3D View. It will be at 0 altitude.

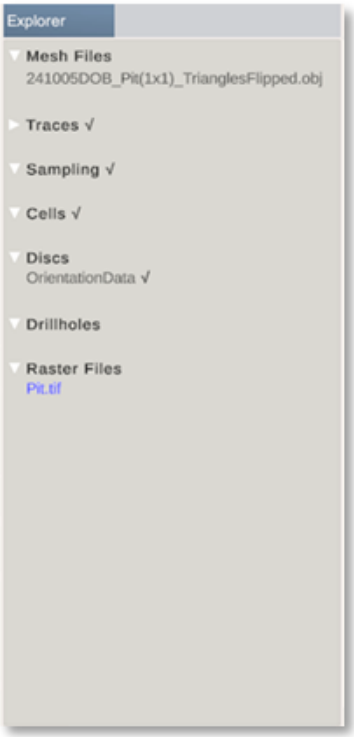
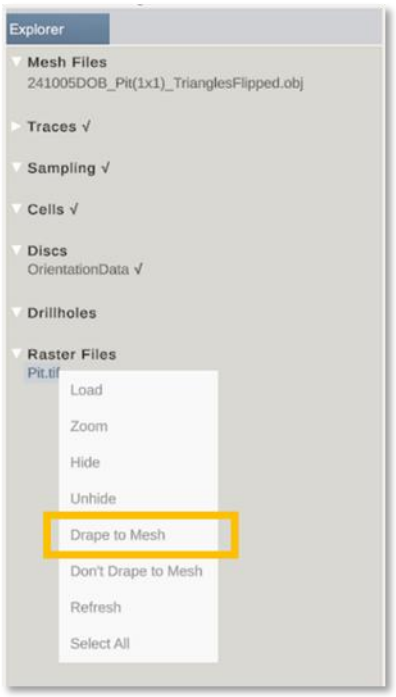


Table of Contents

Step 3.2-18: Raster files can be draped to meshes. Right click the raster file and select Drape to Mesh.



Step 3.2-198: On the Explorer panel everything becomes grayed except the mesh files. Click the mesh that you want to drape the raster file to.

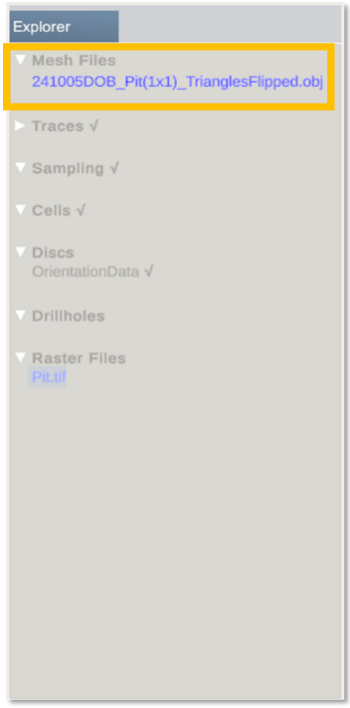
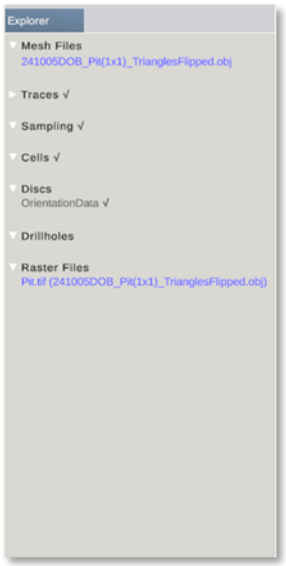


Table of Contents

Step 3.2-29: The name of the mesh file will be displayed next to Raster Files. If the mesh is not loaded. Load the mesh and raster file. Raster files will be displayed on the mesh.



3.3 3D View and View menu

Loaded data are displayed in the 3D View panel.

The Compass is located on the top right corner. It displays the camera orientation of the 3D View.

[Table of Contents]

Using 3D View and View Menu Tools.

An information bar is in the bottom part of the 3D View. When a mouse hovers over an object it displays the related info.

View settings of the 3D View can be changed with the mouse and View menu.

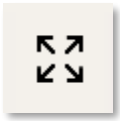
To rotate objects, right click on the mesh and a green spot will appear. This is the rotation axis which is the pivoting center of the image. Press the Ctrl key to rotate the camera, not the objects.

Use the scroll wheel on the mouse to zoom in and out. Zoom direction can be adjusted in the settings.

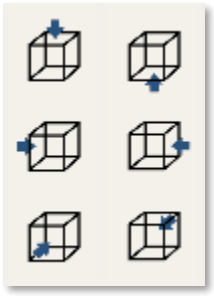
Use the middle mouse button (usually the scroll wheel) to pan the objects.

View menu is located left side of the HiveMap window. View menu is used to adjust the camera of the 3D View and display settings of the objects in the 3D View.

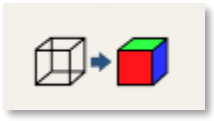
Perspective button switches between perspective and orthographic view. In the orthographic view a scale bar is displayed in the bottom right corner.



Fit to screen button moves the objects to the center and fits hem to the view.



Direction buttons fit the objects to view and rotates the camera to the selected direction.



3.3.1 Mesh Colour

Mesh colour can be changed to show different attributes. These include:

Texture: If the mesh has a texture, it will display the texture. Usually, photogrammetry models are textured mesh files.

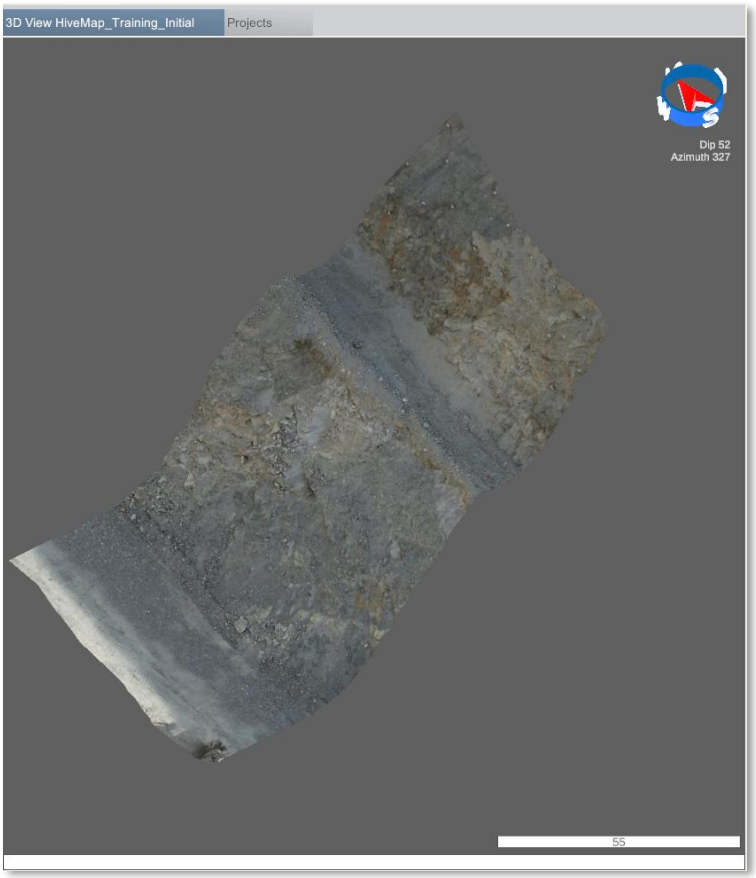
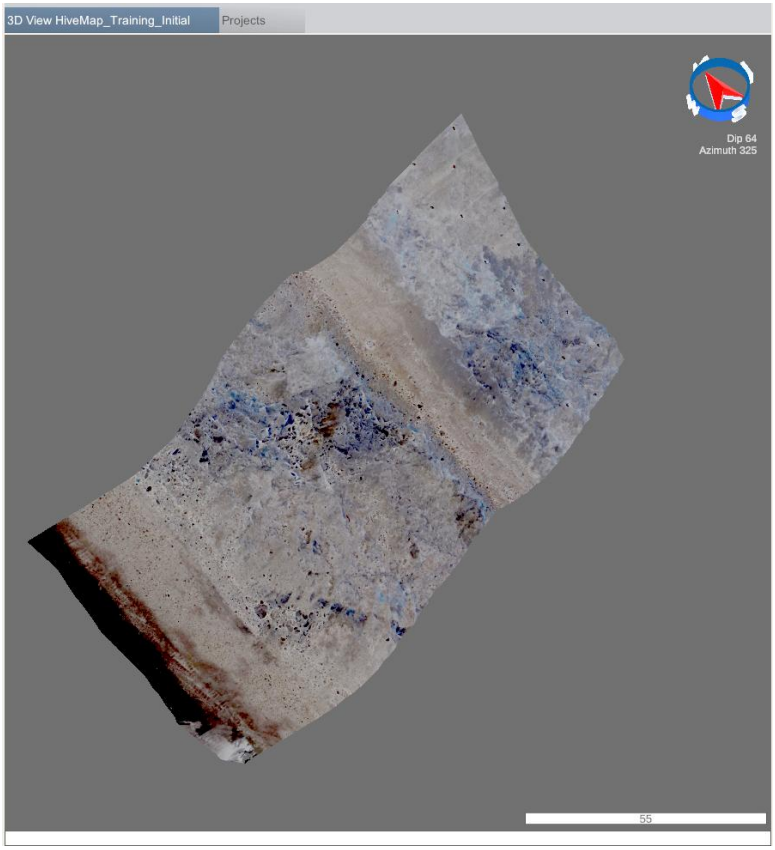


Table of Contents

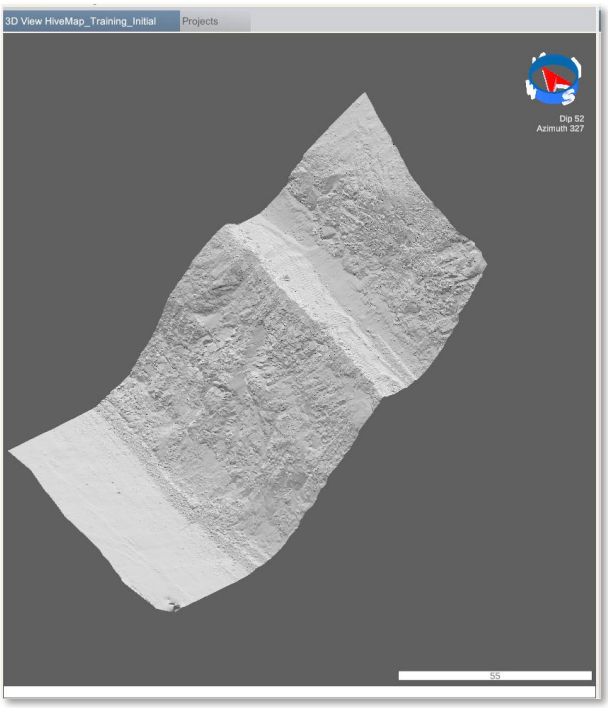
Texture (Inverse Colour): If the mesh has a texture, it will display the texture with inverse colour. This method sometimes can make some structures and features easier to notice.



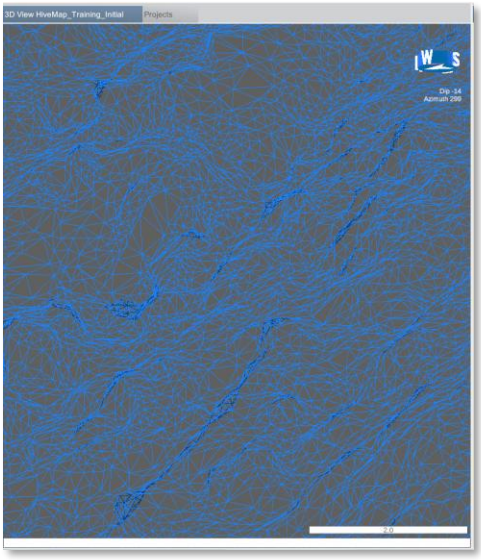
Vertice Colour: If the mesh has colour on its vertices, it will display these colour. When the point clouds converted to meshes each point becomes a vertex and the point colour is transferred to the vertex.

Shade: If there is no texture or vertice colour, this the default view. The surface will be white and shaded according to its vertex normals.

Table of Contents

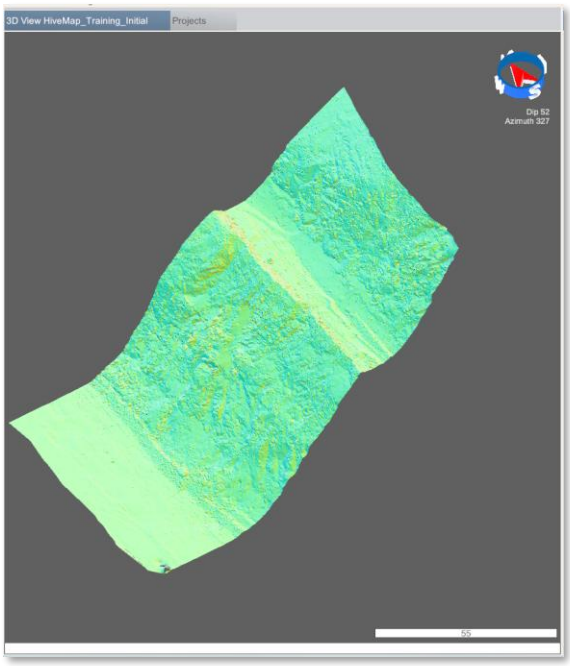


Wireframe: This will show only the triangle edges. This can used to display the mesh geometry resolution and quality.

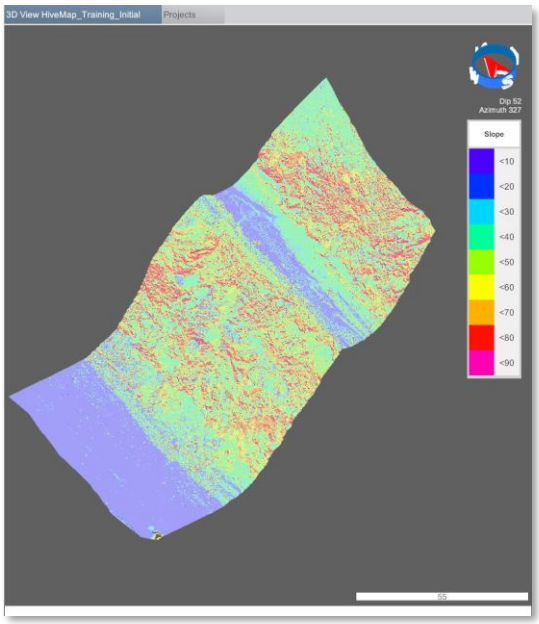


Orientation: This will colourize the surfaces according to the vertex normals. This can be useful to find the joint facets.

Table of Contents

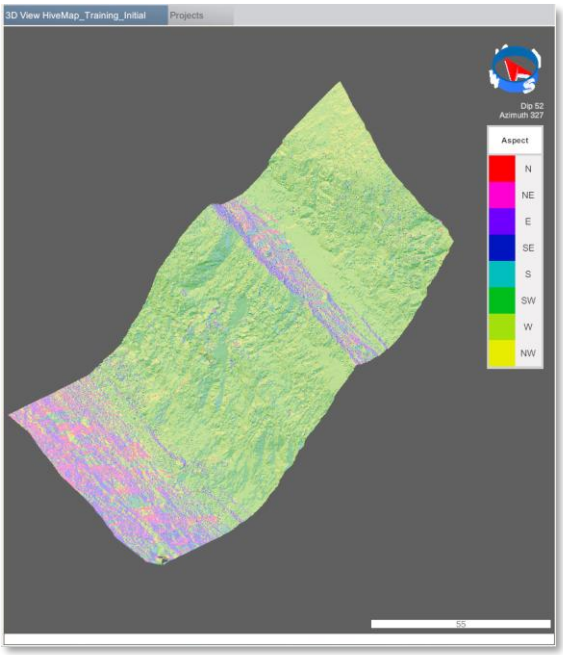


Slope: This will colourize the surfaces according to the Dip of vertex normal. A Slope legend is also displayed.



Aspect: This will colourize the surfaces according to the Dip Direction of vertex normal. An Aspect legend is also displayed.

[Table of Contents]



Elevation: This will colourize the surfaces according to the elevation of the vertices. An elevation legend is also displayed.

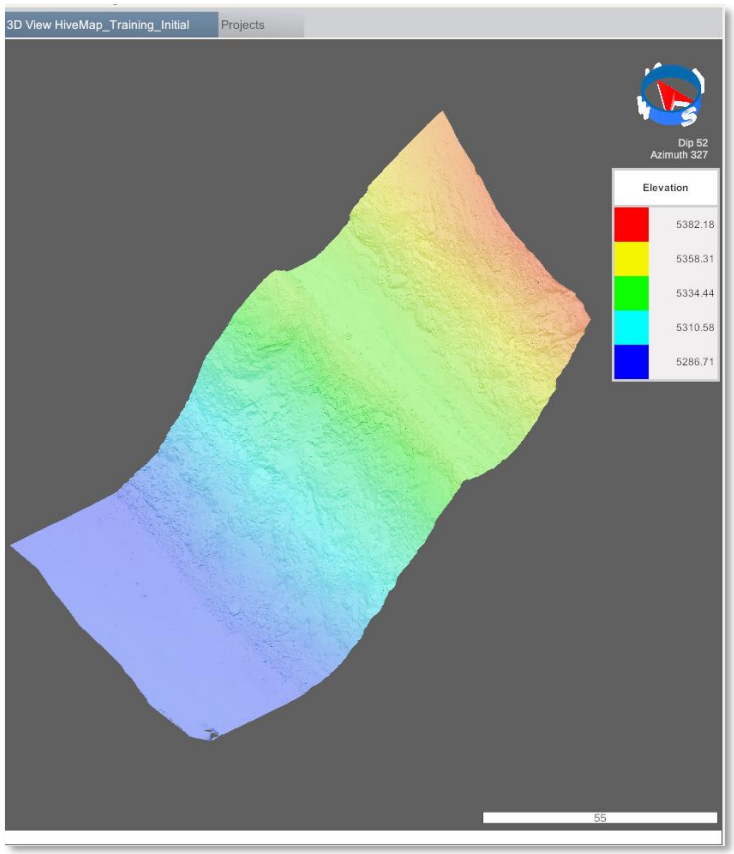
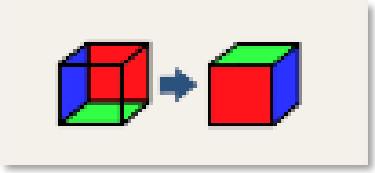


Table of Contents

3.3.2 Mesh View Aspect

Meshes have front and back sides. On default both sides of the meshes are displayed. You can choose to view only one side. This can be important when displaying underground development or tunnels.



Here is a back side culled mesh of a stope, with the inside of the stope visible.

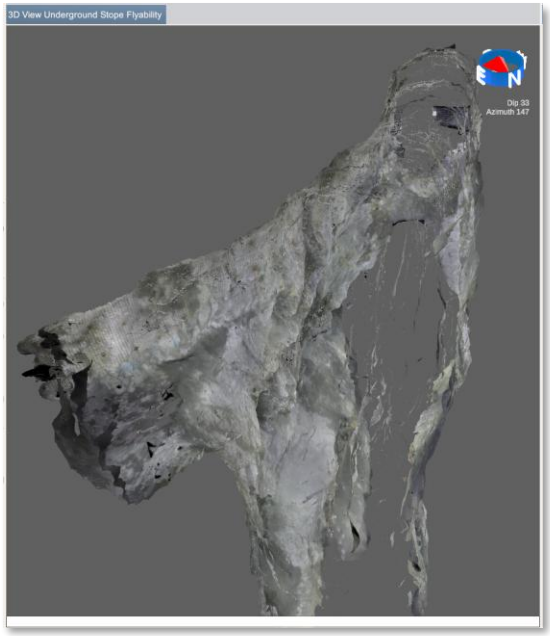
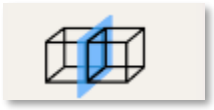


Table of Contents

Here is a front side culled mesh of a stope, with the outside of the stope visible.

3.3.3 Mesh and Data Clipping

The "Clip Mesh and Data" button displays a bounding box around the meshes and hide it if clicked again. When you click on this bounding box, the Properties panel will show buttons for clipping.



The "Resize," "Rotate," and "Reposition" buttons provide controls to adjust the bounding box.

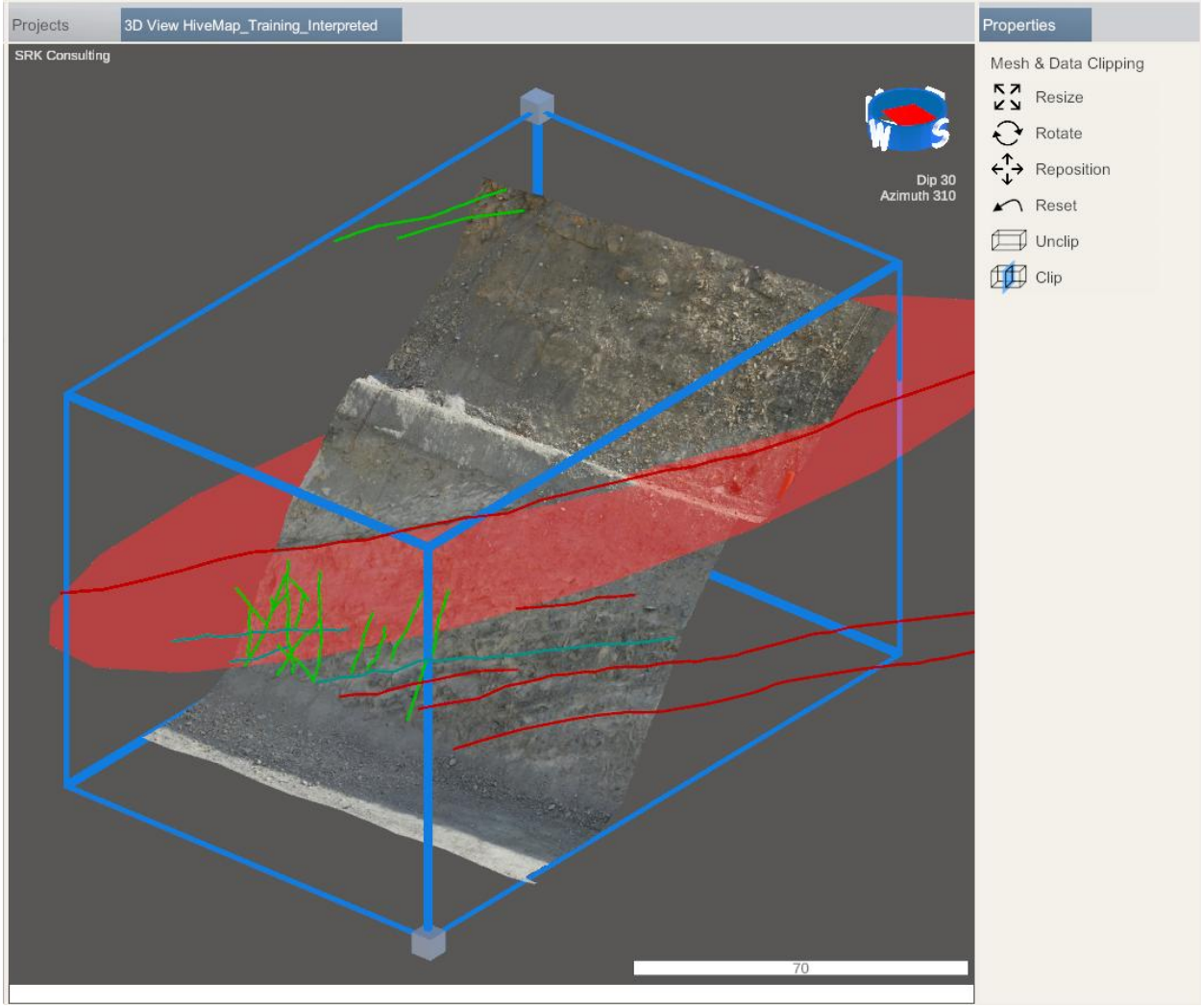
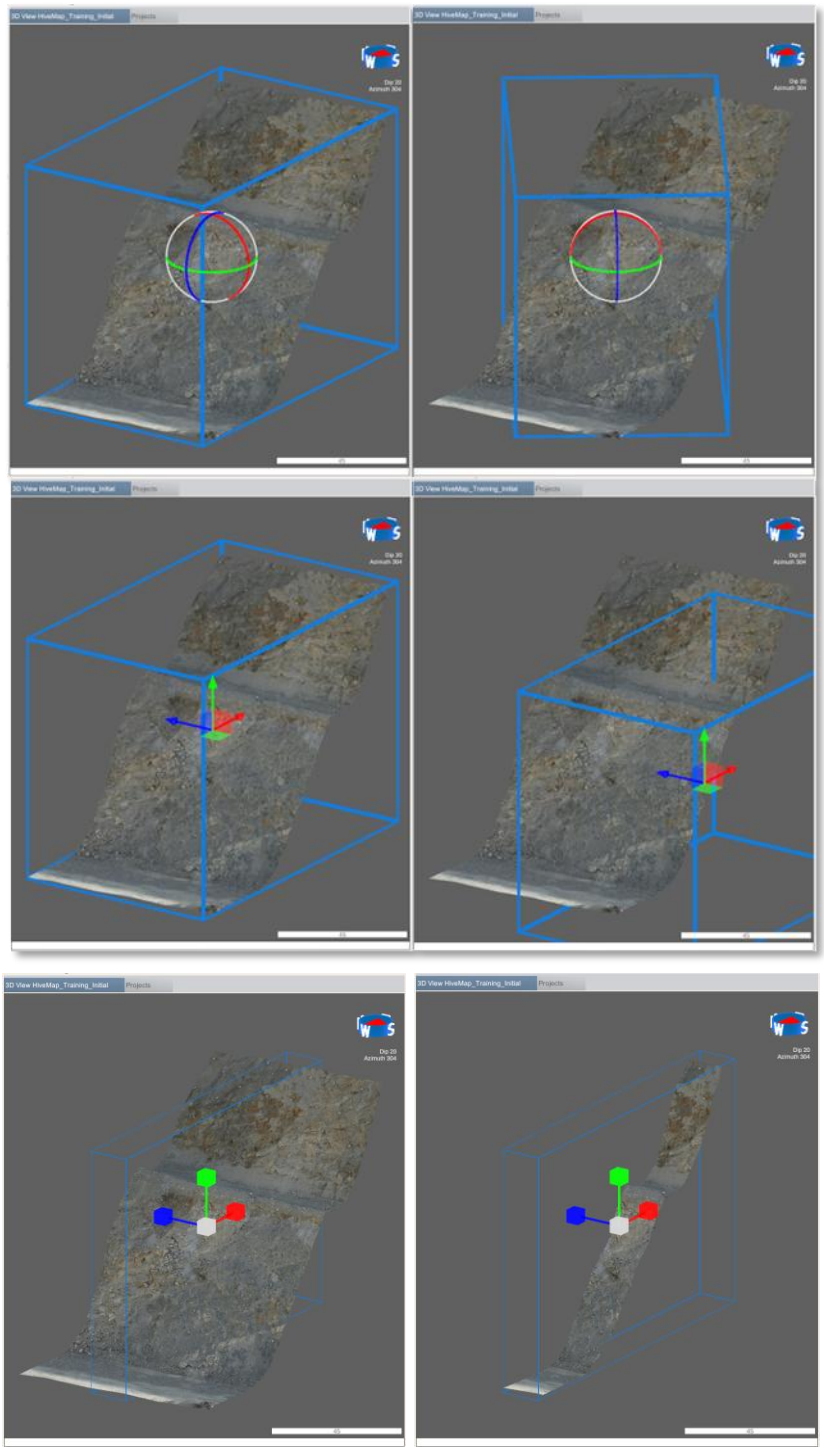
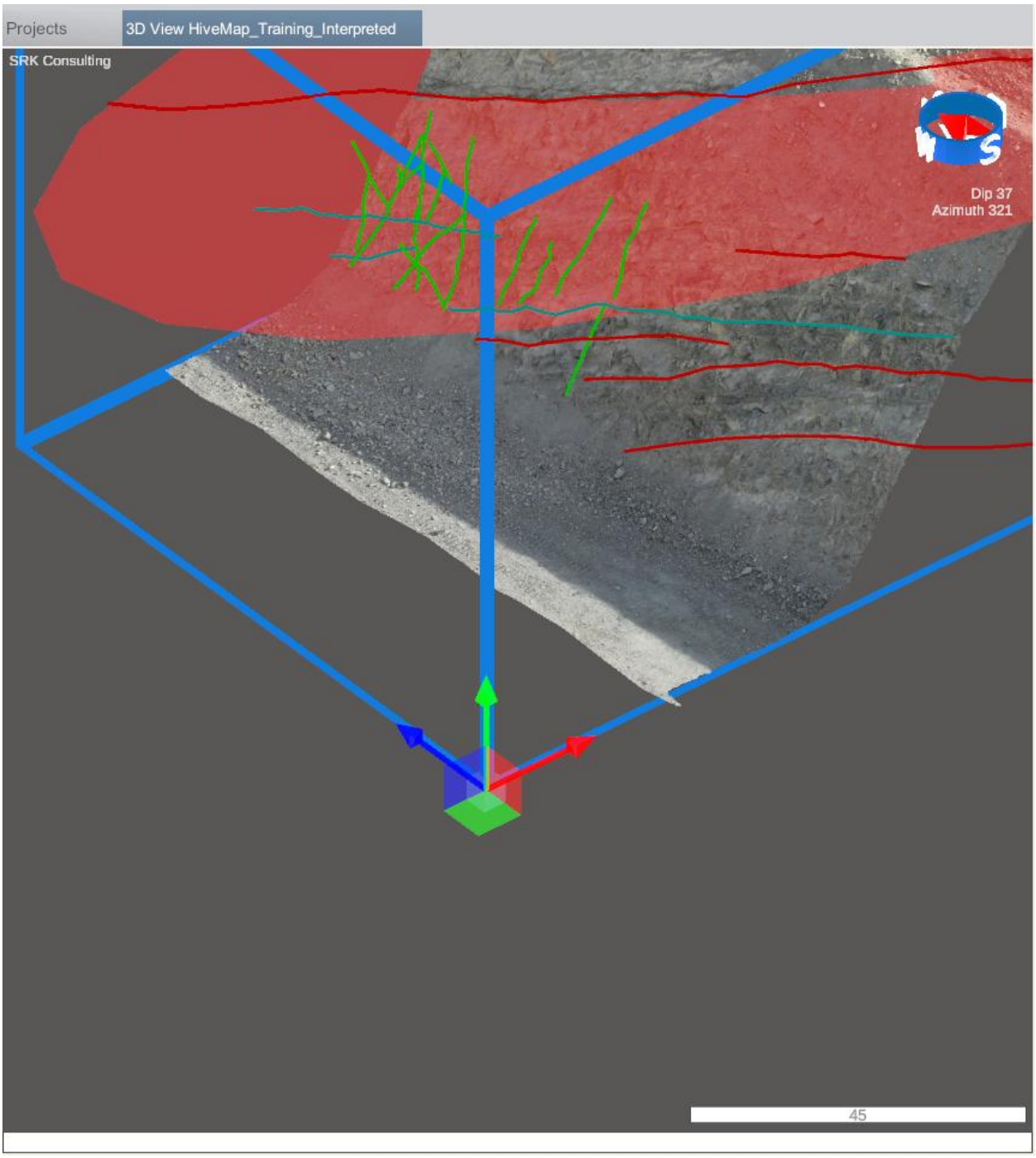


Table of Contents



Bounding box also can be adjusted by using the two boxes which are located on the bottom south-west and top north-west corners. Clicking these boxes will display controls for adjustment.

Table of Contents



Clicking Clip button will hide the mesh and data which is not inside the bounding box. Unclip button unhides the mesh and data. Reset button resets the bounding box.

Table of Contents

3.3.4 Trace Visibility

There are 2 Trace settings in the View Menu.

Width slider adjusts the Trace width.

If the "Always Visible" checkbox is checked, the meshes will not obscure the traces. If unchecked, parts of the trace that are behind the mesh will become invisible.

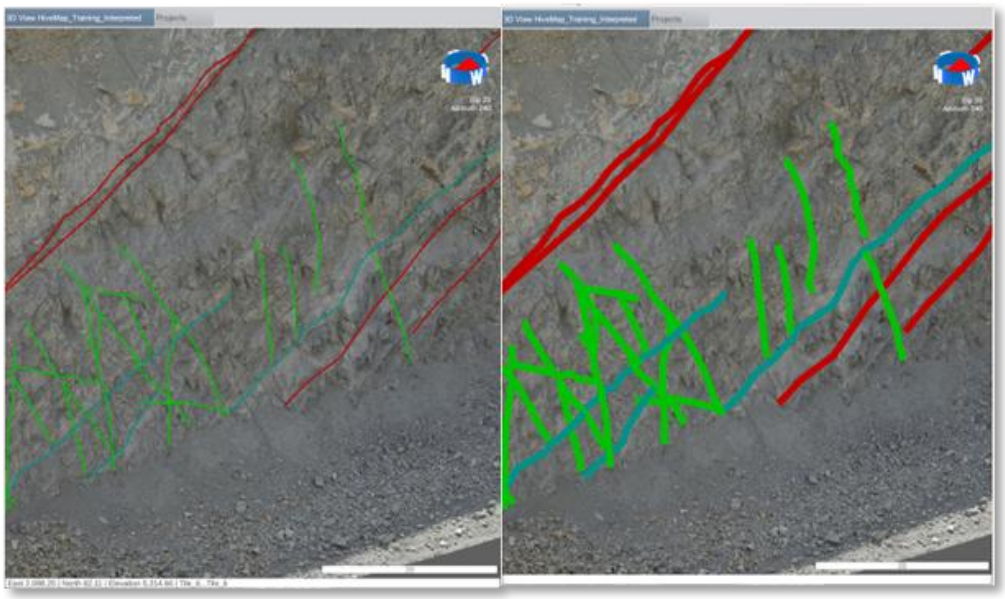
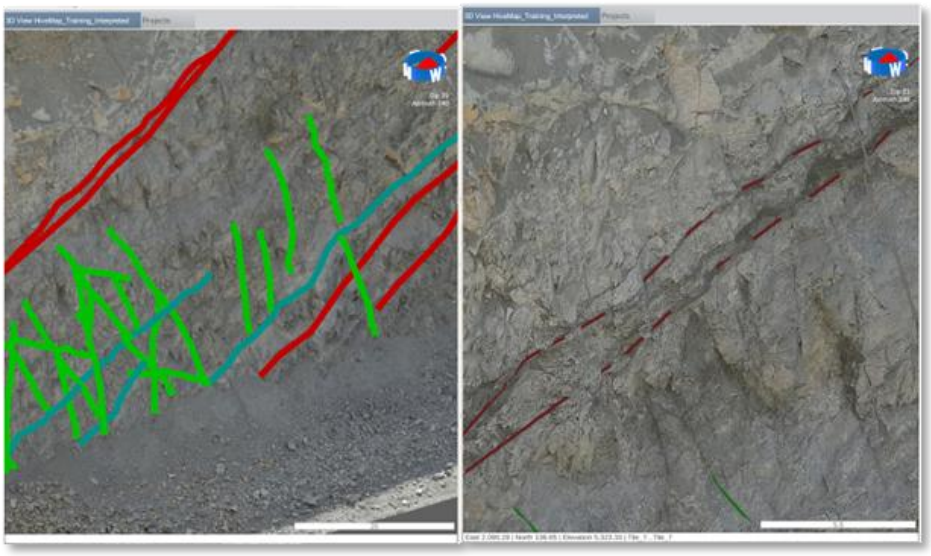


Table of Contents



3.3.5 Disc Visibility

There are 4 Disc settings in the View Menu.

The Thickness slider adjusts the thickness of the discs.

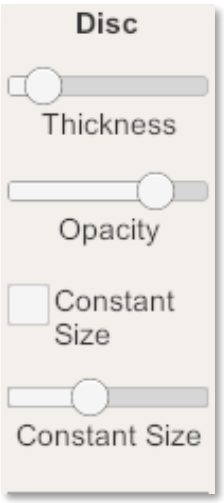
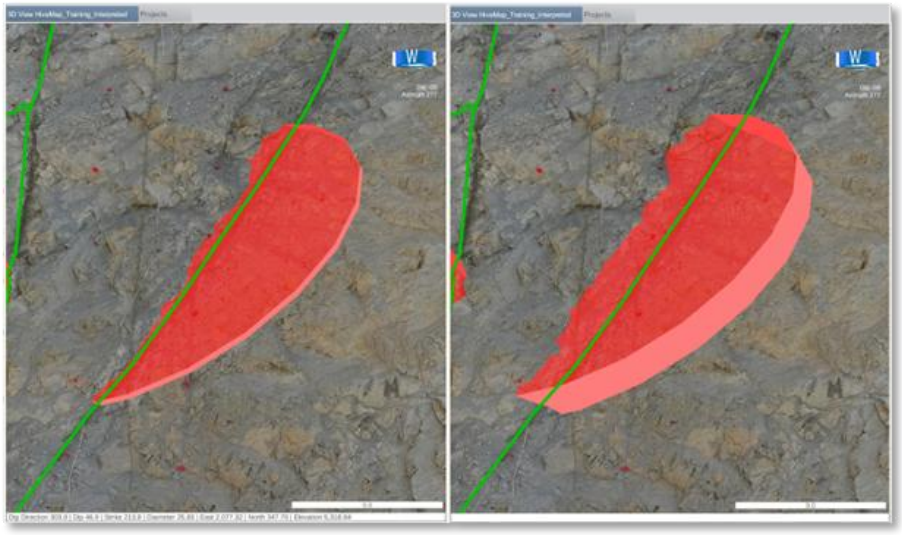
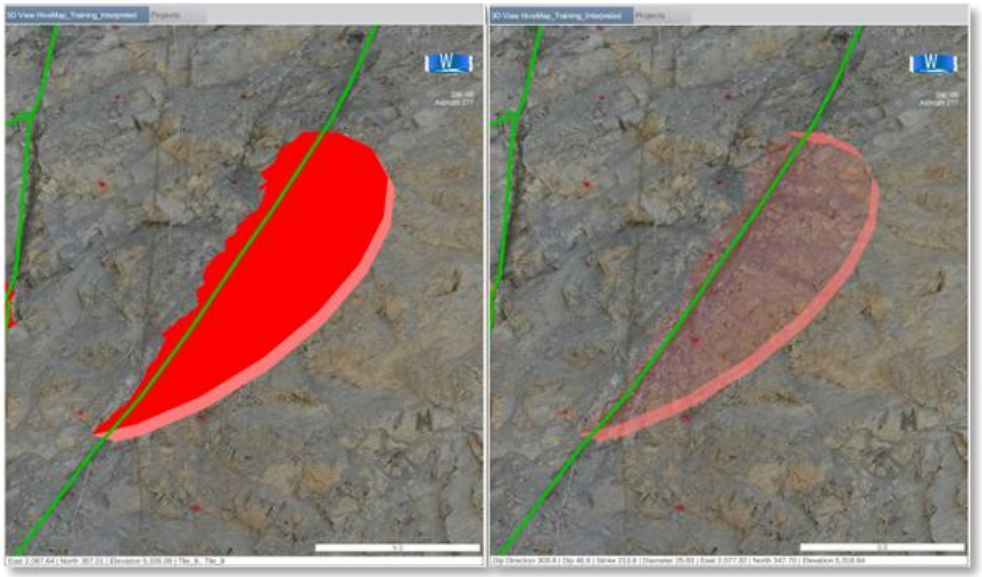


Table of Contents

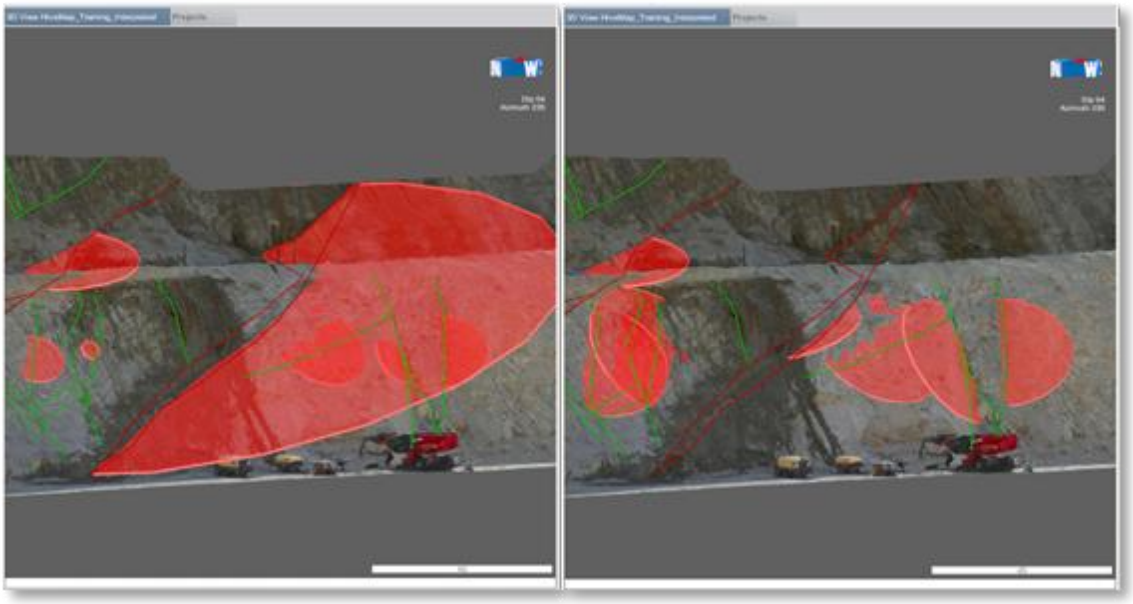


The Opacity slider adjusts the transparency of the discs.



If the Constant Size check box is checked all the discs are displayed with the same diameter relative to the screen. This diameter can be adjusted with the Constant Size slider.

Table of Contents



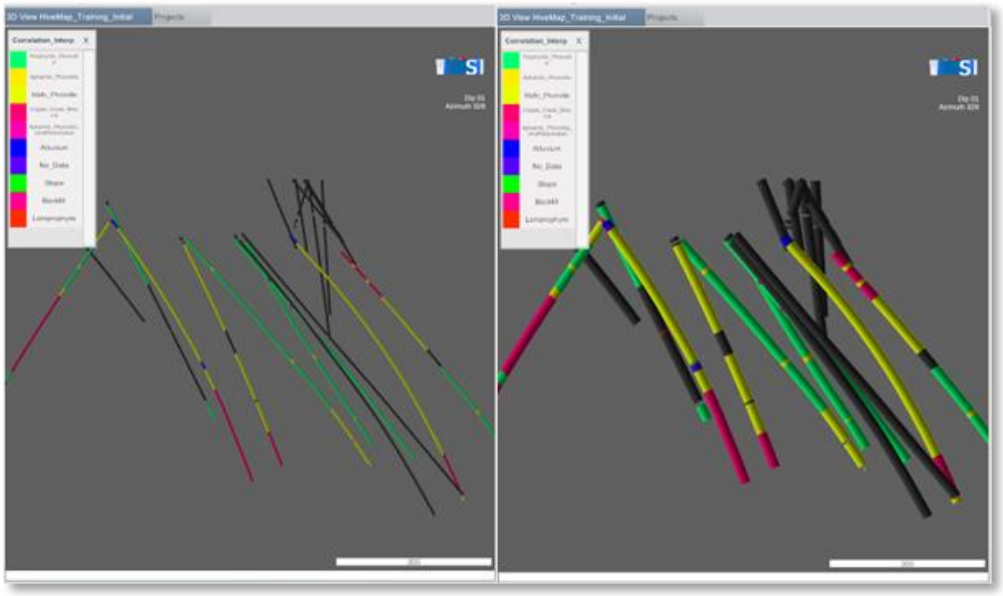
3.3.6 Drillhole Visibility

There are 3 Drillhole settings in the View Menu.

Display Drillholes checkbox hides or unhides the Drillholes.

The Width slider adjusts the drillhole diameter.

Table of Contents



The "Toggle Drillhole Attributes" button allows you to switch between different attributes and interval files. The first text below this button displays the currently used interval file, while the second text shows the attribute name.

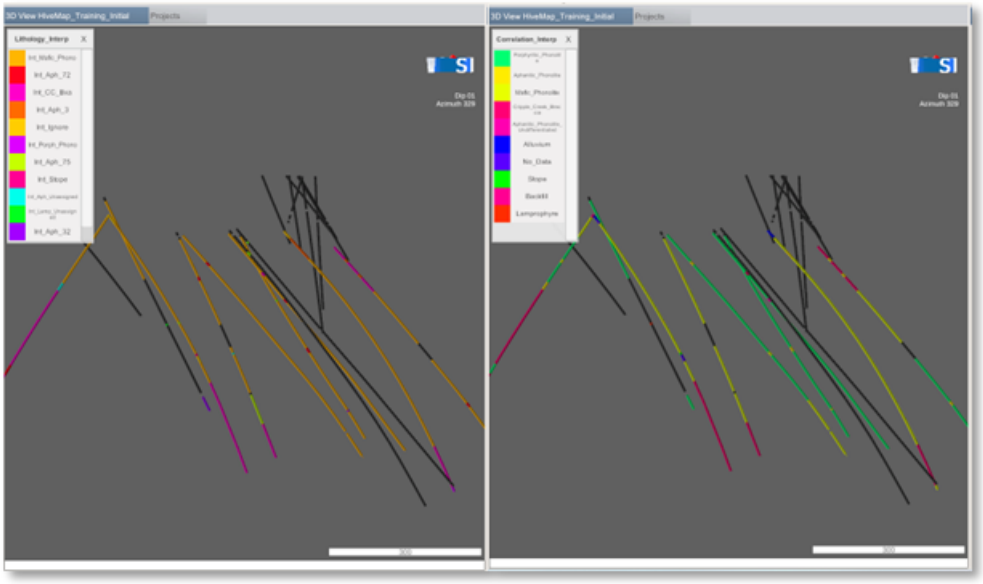


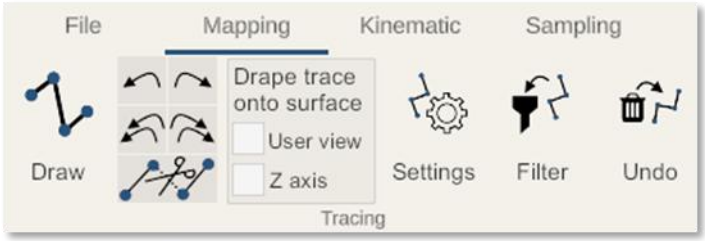
Table of Contents

3.4 Mapping

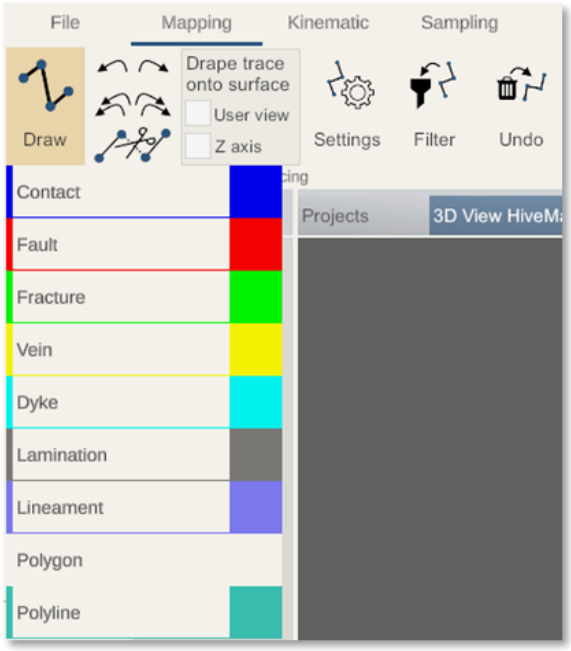
The Mapping menu has 4 tools for mapping: Tracing, Measurement Orientation Tool, Stereonet, and Joint Sets.

3.4.1 Tracing

The tracing tool group enables users to trace the geological structures.

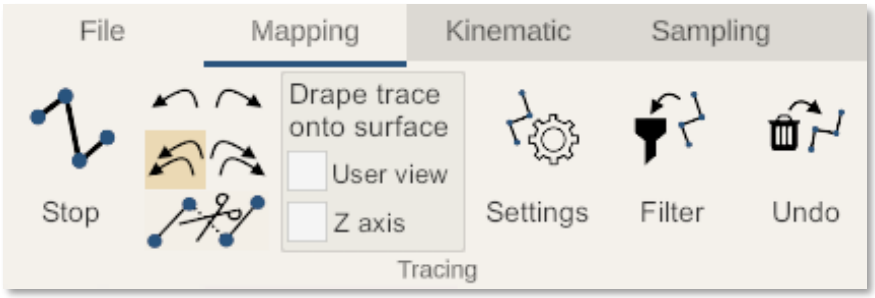


Draw button opens the list of structures.



Selecting a structure starts the tracing process. During this process other tools remain disabled.

Clicking on Stop, stops the process and will allow for other tools to be used.



Use your left mouse start tracing the structure on the mesh. You can do single clicks or hold down the left mouse button.

The "Drape Trace onto Surface" option drapes the trace onto the surface. If the "User View" checkbox is checked, the draping direction can be adjusted with the mouse. If the "Z Axis" checkbox is selected, the trace will be draped vertically.

With the draping feature, you can capture a mesh shape with just two mouse clicks.

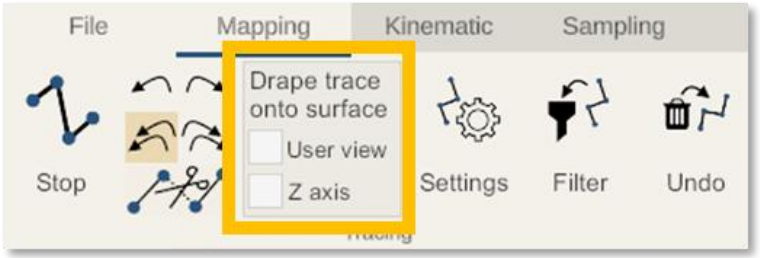
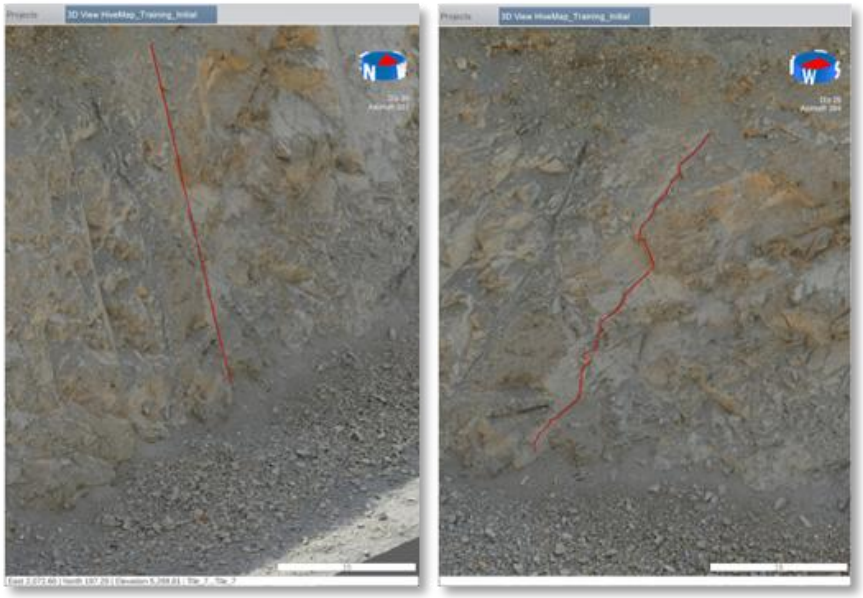


Table of Contents



Clicking the ‘Undo button’ or backspace key will delete the last node. Clicking on the ‘Redo button’ brings back the removed nodes.
The ‘Undo last click button’ will delete all the nodes which have been created with a single click. The ‘Redo last click ‘brings back all of the removed nodes that had been created with single clicks.

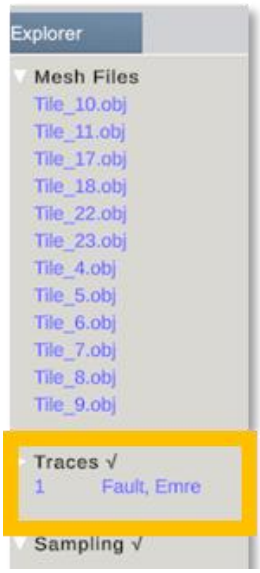
Table of Contents

While tracing, you can split traces to follow structures that are not continuous.

After finishing tracing click on the ‘Stop button’ to end the process.

Instead of clicking on the Stop button, you can press the ‘Enter Key’ or click with the right and left mouse buttons simultaneously. This will end and restart tracing for the same structure.

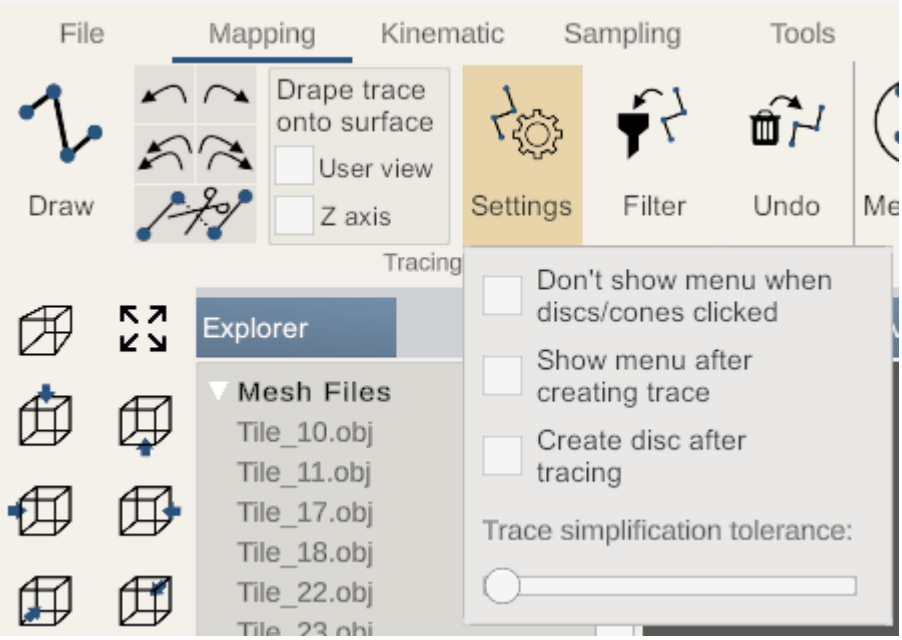
The new trace file is displayed on the Explorer panel under the Traces branch. Trace files are saved to the Traces folder inside the Project folder.



[Table of Contents]

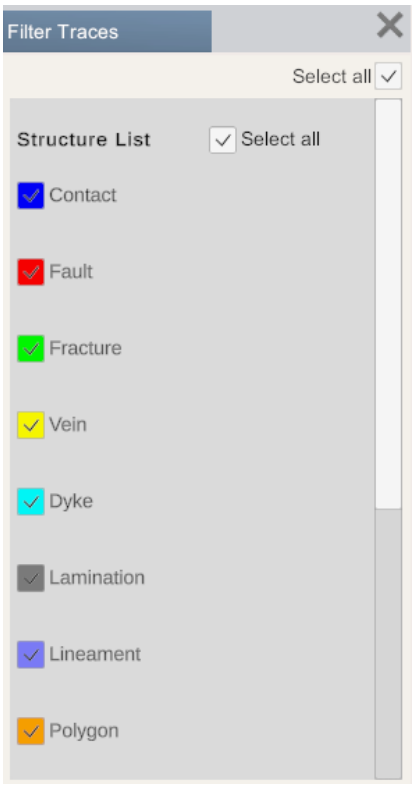
The Trace Settings menu has 4 options:

1. Don’t show menu when disc/cones clicked: When the trace discs and cones are clicked the trace menu is displayed. If this option is unchecked, only clicking the trace itself will open the trace menu.
2. Show menu after creating trace: If this option is checked, after finishing tracing, the trace menu will be displayed automatically.
3. Create disc after tracing: This will automatically create the trace disc/cone after the tracing is finished.
4. Trace simplification tolerance: The simplification process leverages the [Ramer-Douglas-Peucker algorithm] to decrease the number of points, guided by the Tolerance parameter.

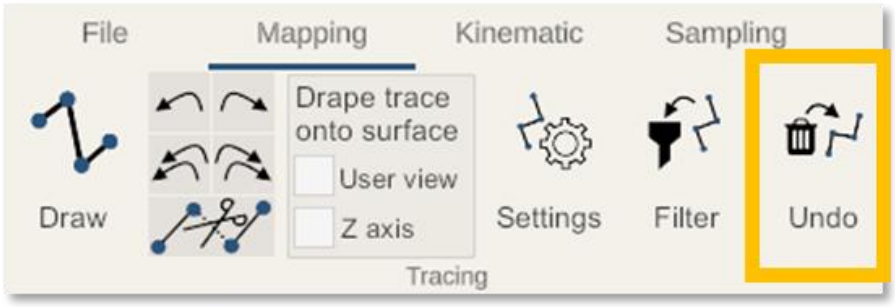


The "Trace Filter" button opens the trace filtering menu, allowing users to filter the loaded traces based on structure, trace author, and mesh.

Table of Contents

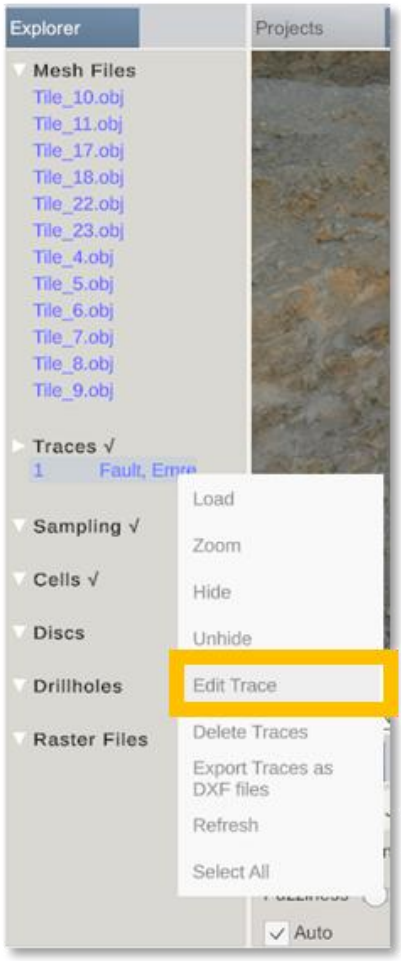


The "Undo" button reverses the most recent trace deletion. Deleted traces are stored in the Recycle Bin within the project folder.

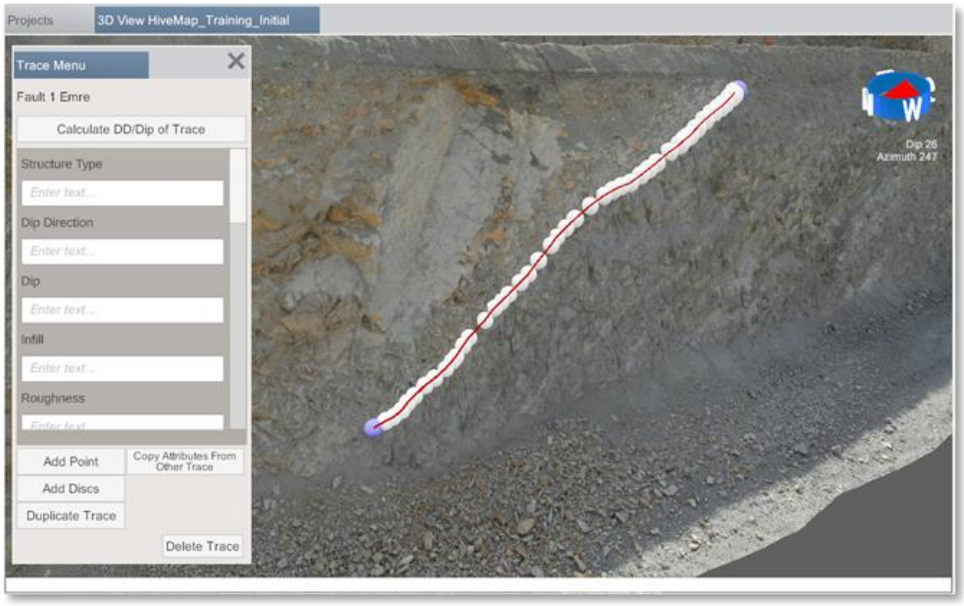


To edit the trace file, left click the trace in the 3D View menu. Alternatively, right click the trace on the Explorer panel and then the Edit Trace on the right click menu.

Table of Contents

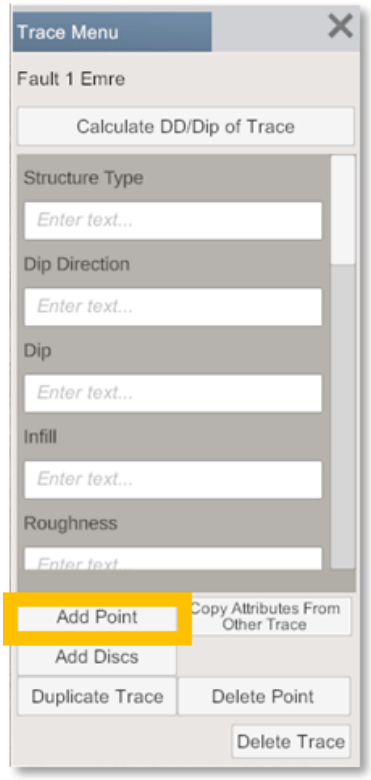


The Trace Menu is displayed, and the nodes of the trace polyline/polygon becomes visible.



[Table of Contents]

Click Add Point button and then click the trace line.



A new node will be added where the line is clicked.

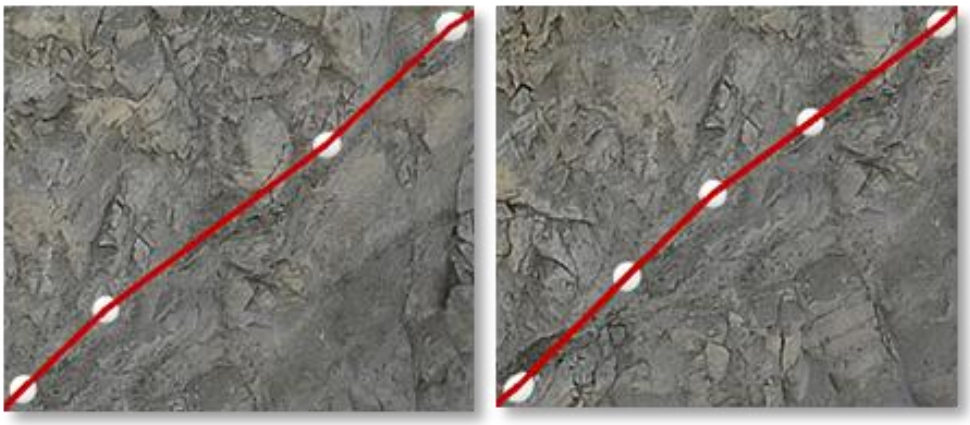
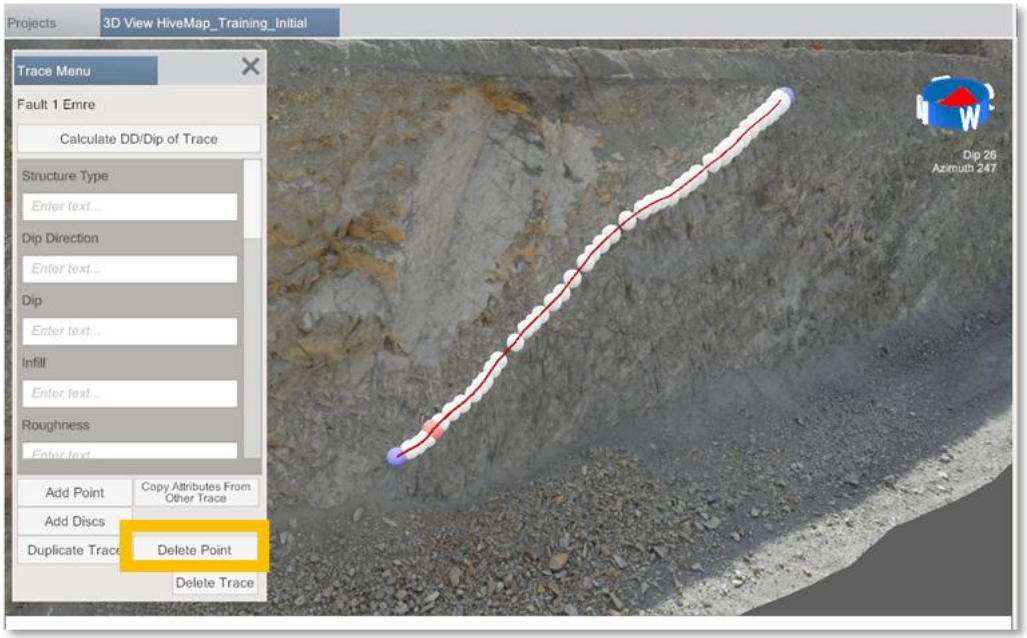
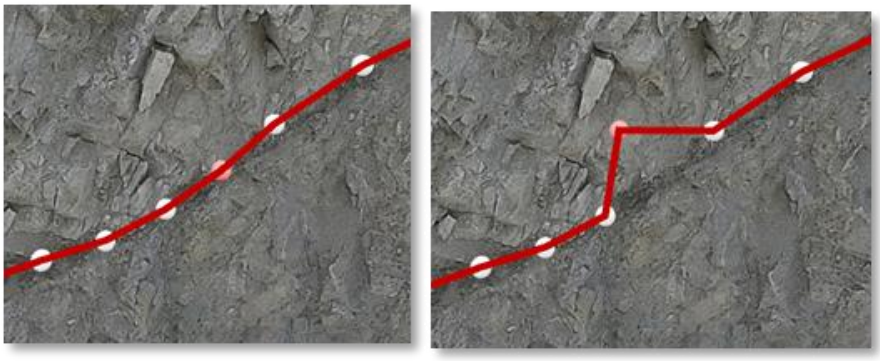


Table of Contents

Click one of the white nodes. The node will become red, and on the Trace Menu, the Delete Point button will become visible. Click this button to delete the selected red node.

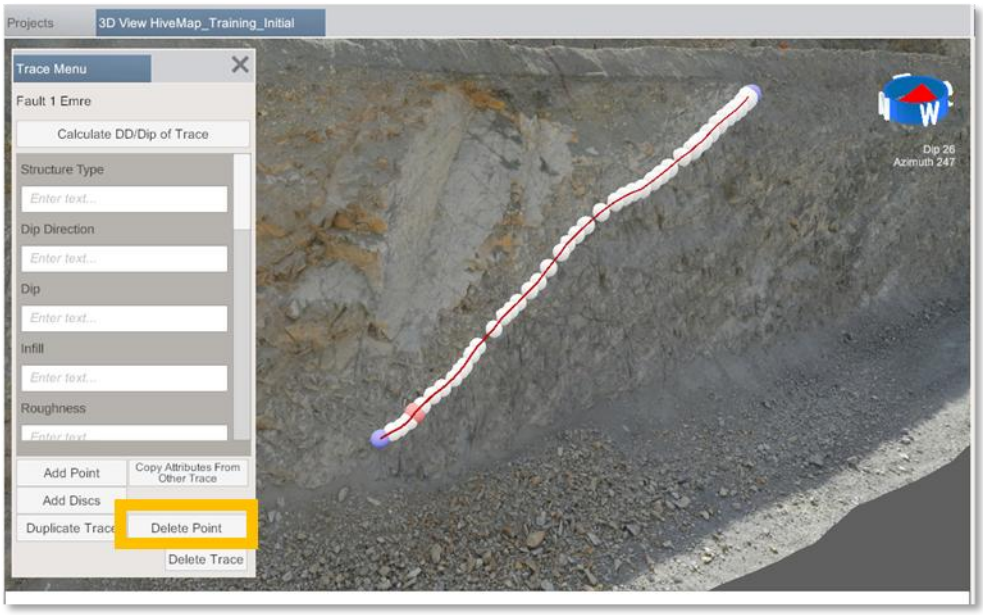


Click the mesh and the red node will move to the clicked position on the mesh.



Click the first or last node, which will be in purple. The node will turn red, and the "Delete Point" and "Continue Tracing" buttons will appear in the Trace Menu. Click "Continue Tracing" to close the Trace Menu and hide the nodes, allowing tracing to resume from the selected node.

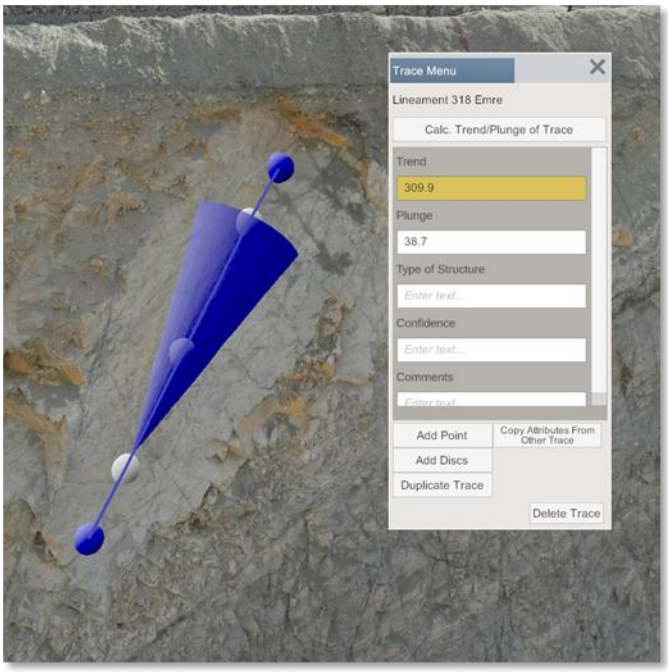
Table of Contents



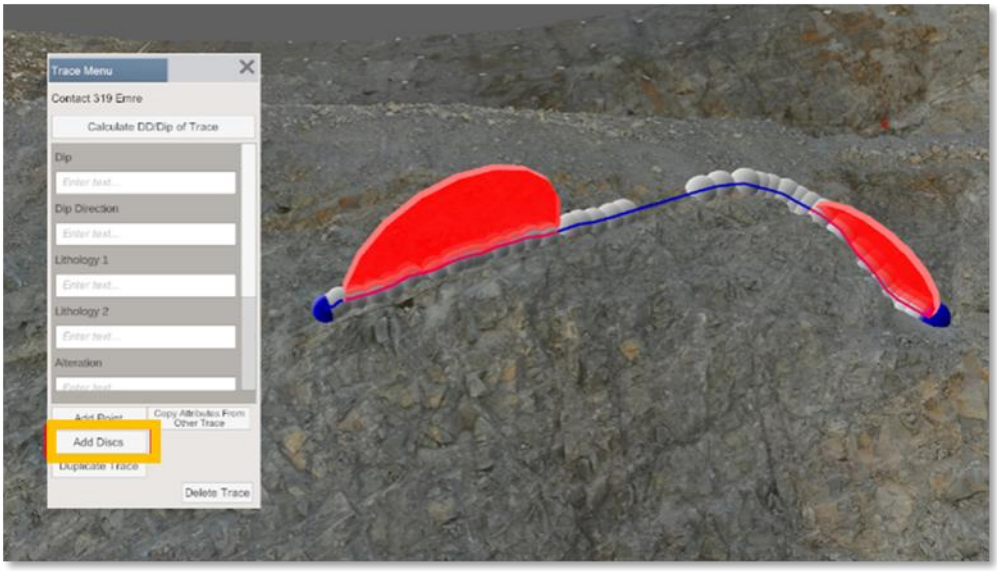
An orientation can be calculated for each trace line by selecting the trace and pressing the *Calculate DD/Dip of Trace.*

[Table of Contents]

If it is a linear structure, you can fit a cone. This may be done by selecting the *Lineation* trace option.

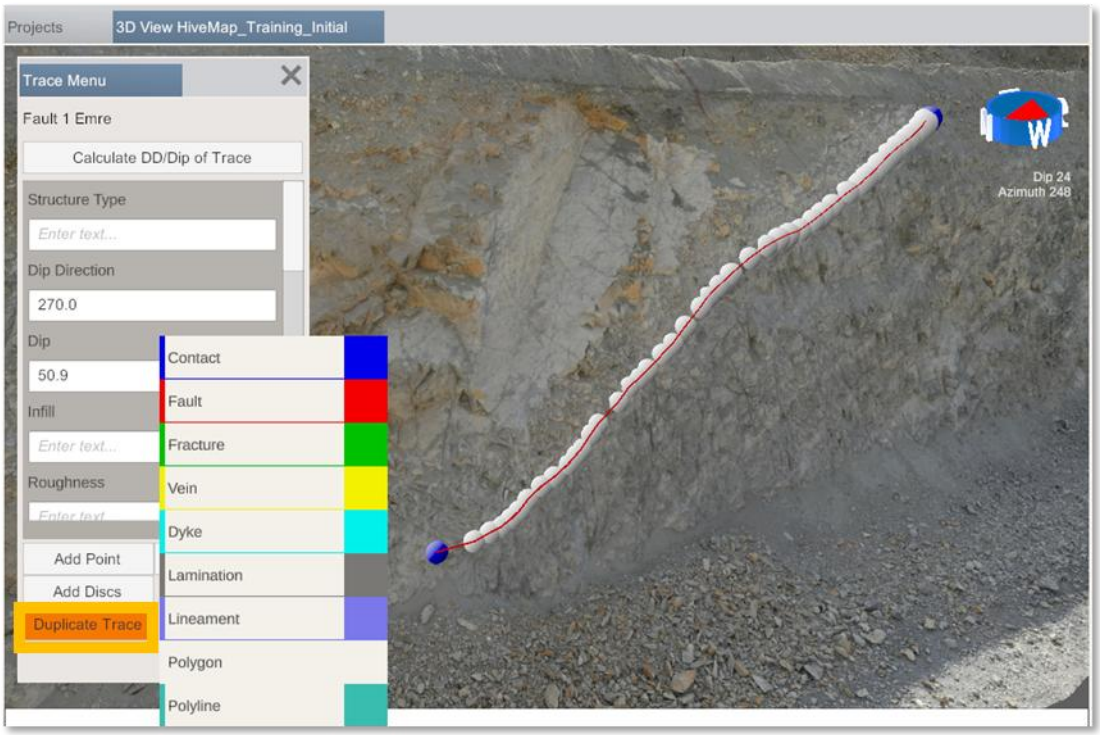


If it is a folded structure, you can add discs along the trace to capture an orientation change.

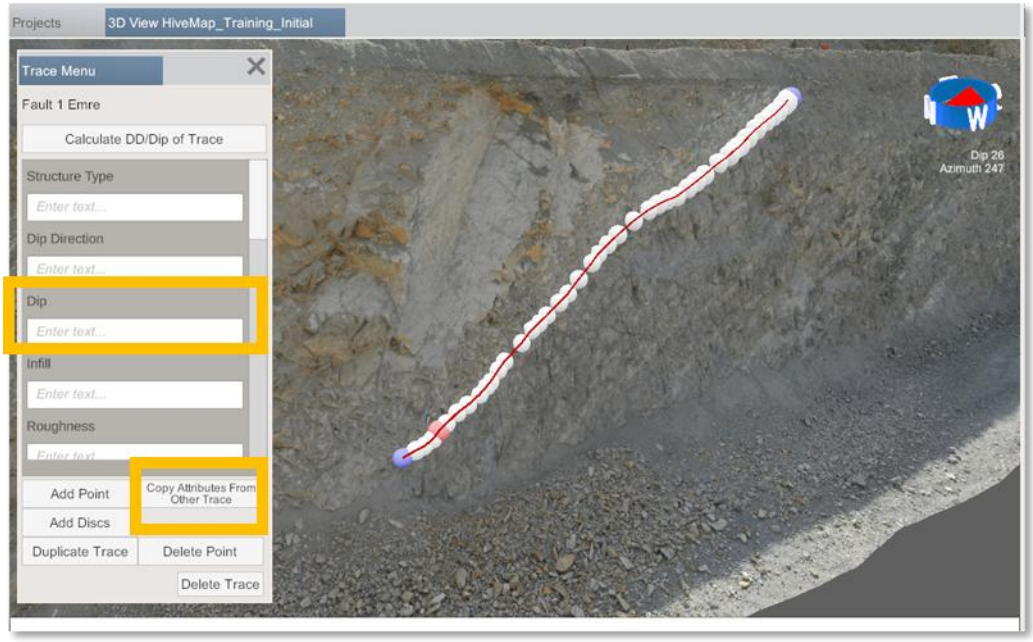


Duplicate trace button duplicates the trace as a different structure. For example, if a fault is a contact, after tracing the fault it can be duplicated as contact.

[Table of Contents]



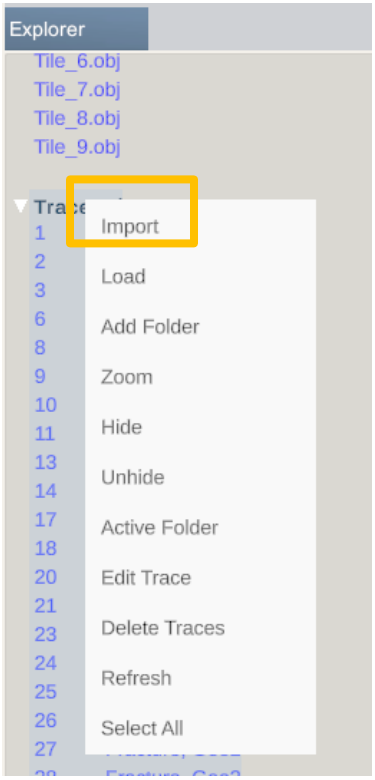
"Copy Attributes from Other Trace" will copy the form data from another trace. After clicking this button, select the source trace that corresponds to the same type of structure.



[Table of Contents]

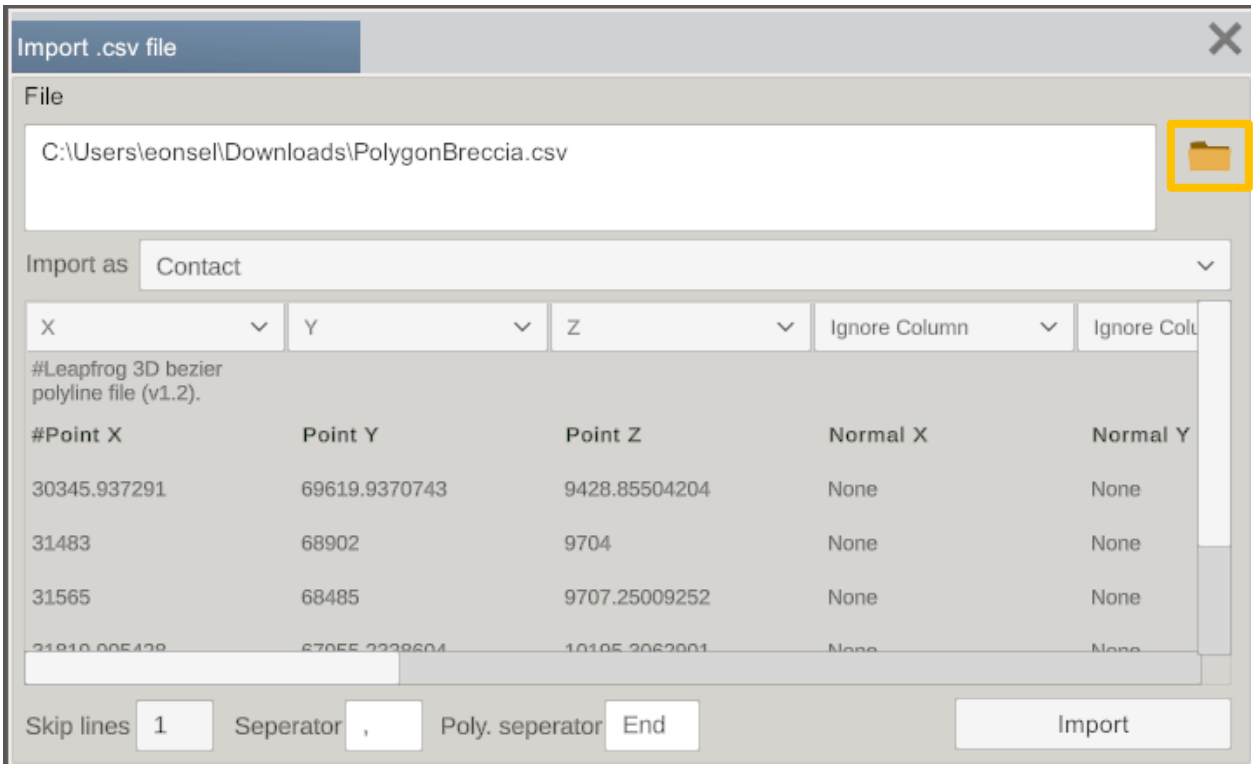
In the Explorer panel, the right-click menu includes options to Zoom, Hide, Unhide, Edit, and Delete traces.

In the Explorer panel, right click menu for the Traces branch includes the Import button, which will display .csv import menu.

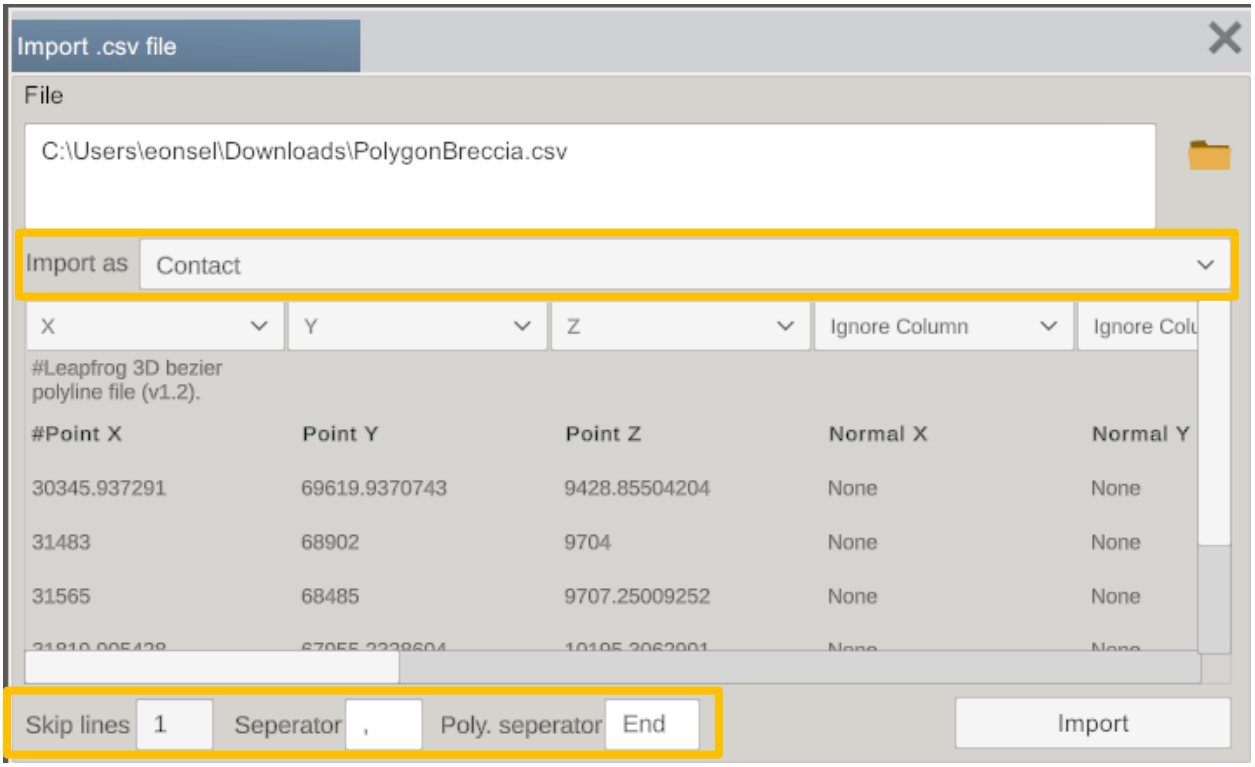


[Table of Contents]

Click the folder button to select the .csv file.

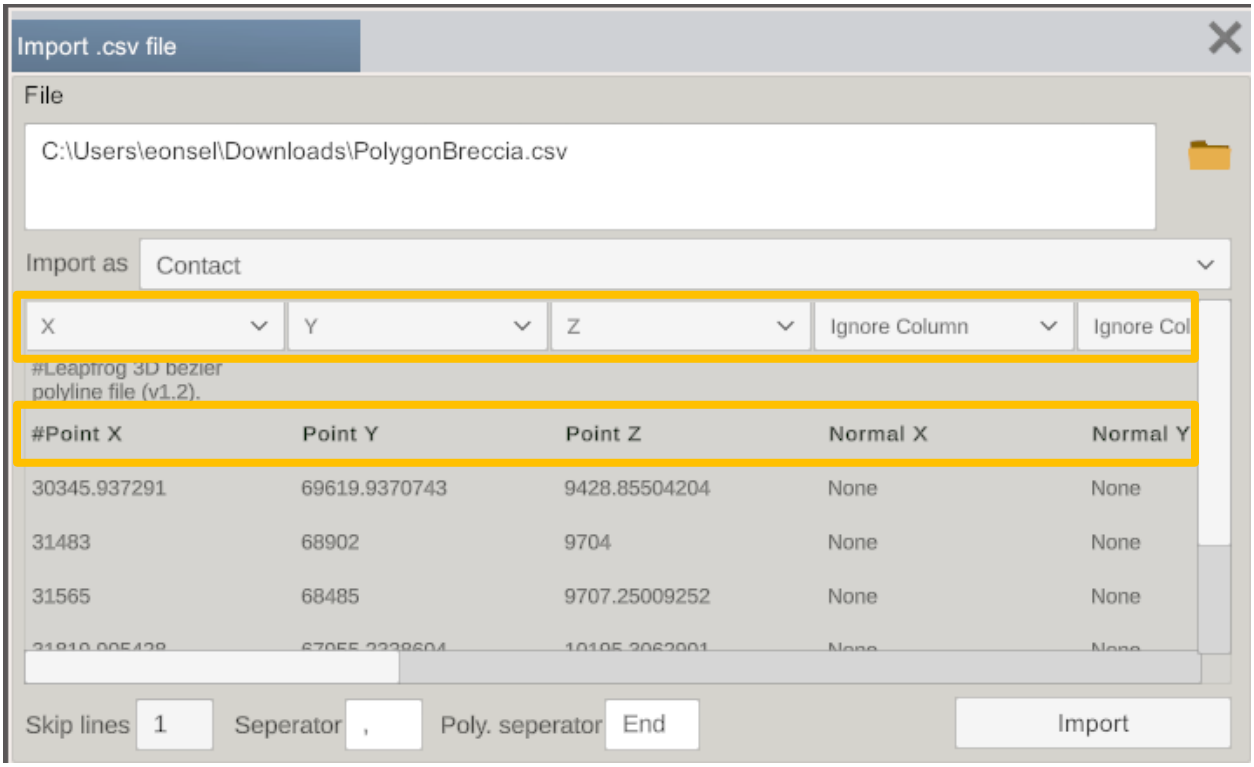


Select the structure type that the polyline is going to represent and adjust the “”skip lines”, “seperator” and “polyline seperator” parameters to correctly parse the .csv file.



[Table of Contents]

Select the matching columns for x, y and z. Click import to create the trace file.

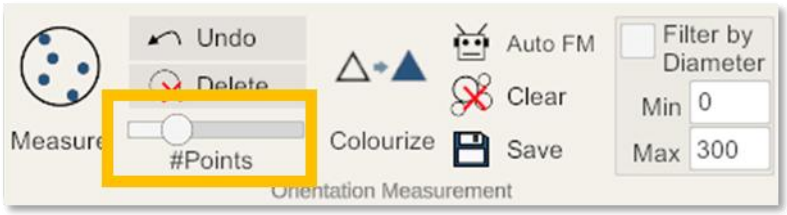


3.4.2 Orientation Measurement Tool

The orientation measurement tool enables users to measure orientation of the geological structures.

To begin the process, navigate to the Mapping tab on the ribbon and click the "Measure" button in the Orientation Measurement group.

The number of sampling points can be adjusted using the Points slider. Increasing the number of sampling points will provide a more accurate orientation measurement. The minimum value is 3, and the maximum value is 12.

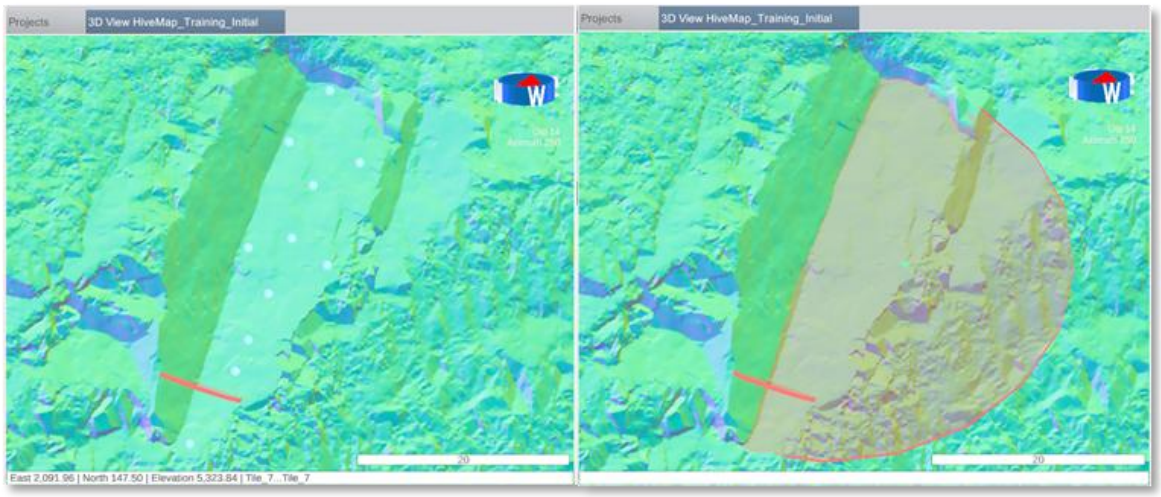


Sampling points can be placed on the structure traces and facets:

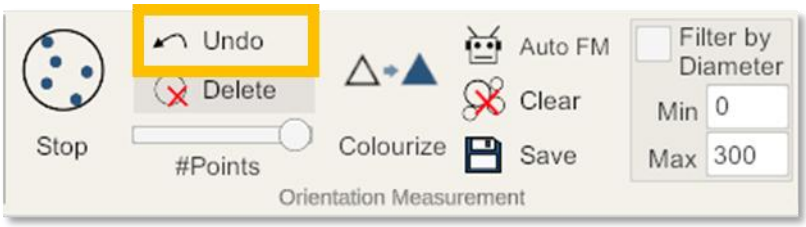
1. Identify a joint trace and begin placing sampling points on it using your left mouse button. A disc will form once you've placed the number of points specified by the point slider. The orientation of the disc is determined by calculating the best fitting plane. The disc's location and size are based on the two points that are furthest apart.

Table of Contents

2. Locate a facet of a joint on the mesh. Use your left mouse button to place sampling points on the facet surface. A disc will form once you've placed the number of points specified by the point slider.



While placing sampling points you can press the backspace key or click the Undo button to undo the last sampling point.



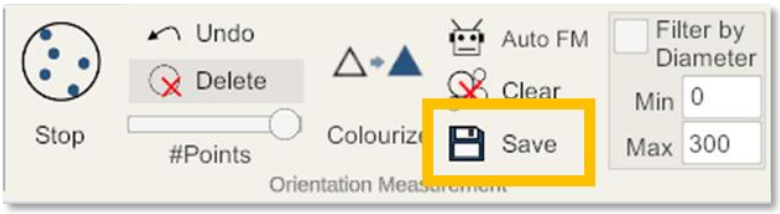
If you have flat facets to map, you can adjust the point slider to minimum and create discs quickly by just placing 3 points for each of them.

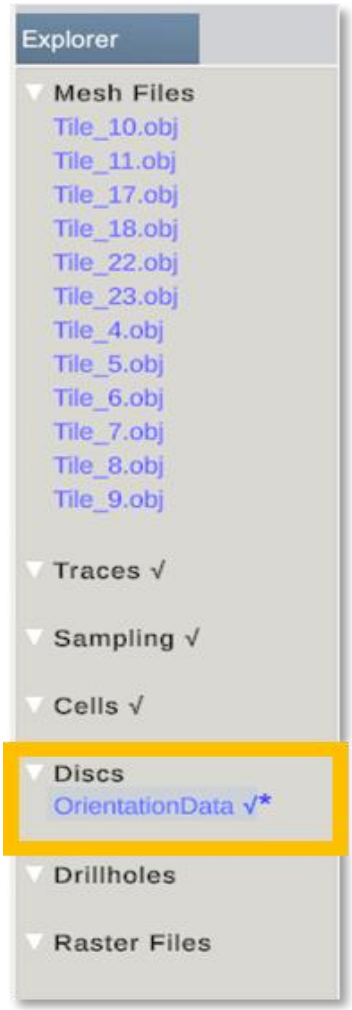
Alternatively, you can keep the Points slider at maximum and press the enter key to create discs by using variable number of points for each joint. If you have facets and traces with rough, wavey surfaces you need to increase the number of sampling points to get the correct average orientation of the structure.

After you've finished mapping, click the Stop button.

Table of Contents

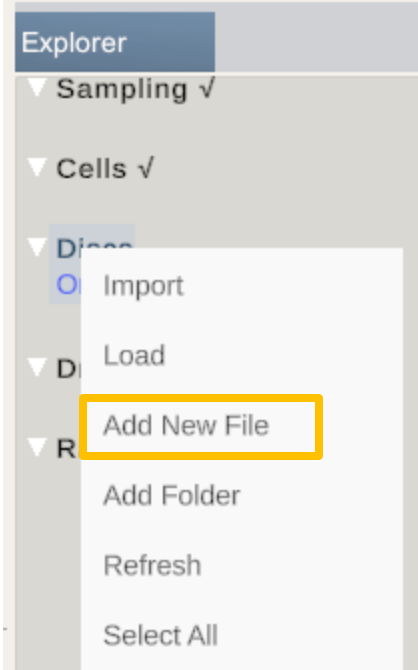
Your disc data will be saved to OrientationData.csv file in the Project/OrientationData folder. If there is any unsaved data, a * sign is displayed next to the file name. Click the Save or Stop button to save data to the file and remove the * sign.



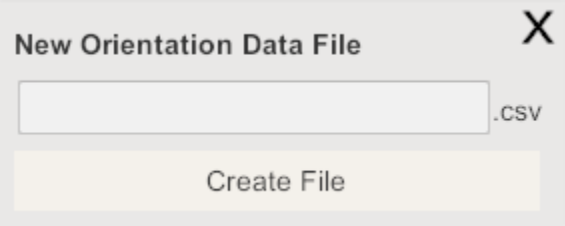


If you want to save into a different file, right click the Discs branch in the Explorer panel and click add new file button.

[Table of Contents]

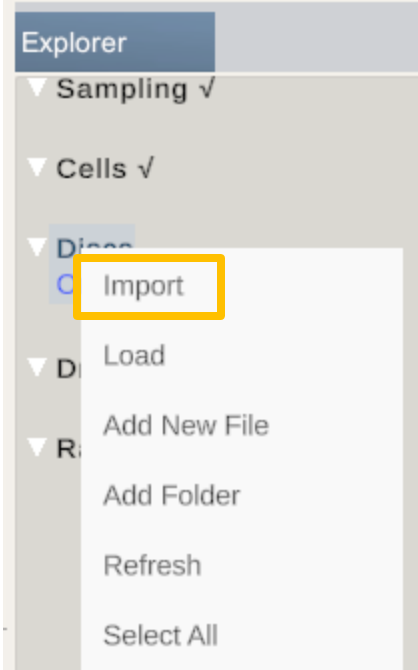


Type the name of the file and click the Create File button.

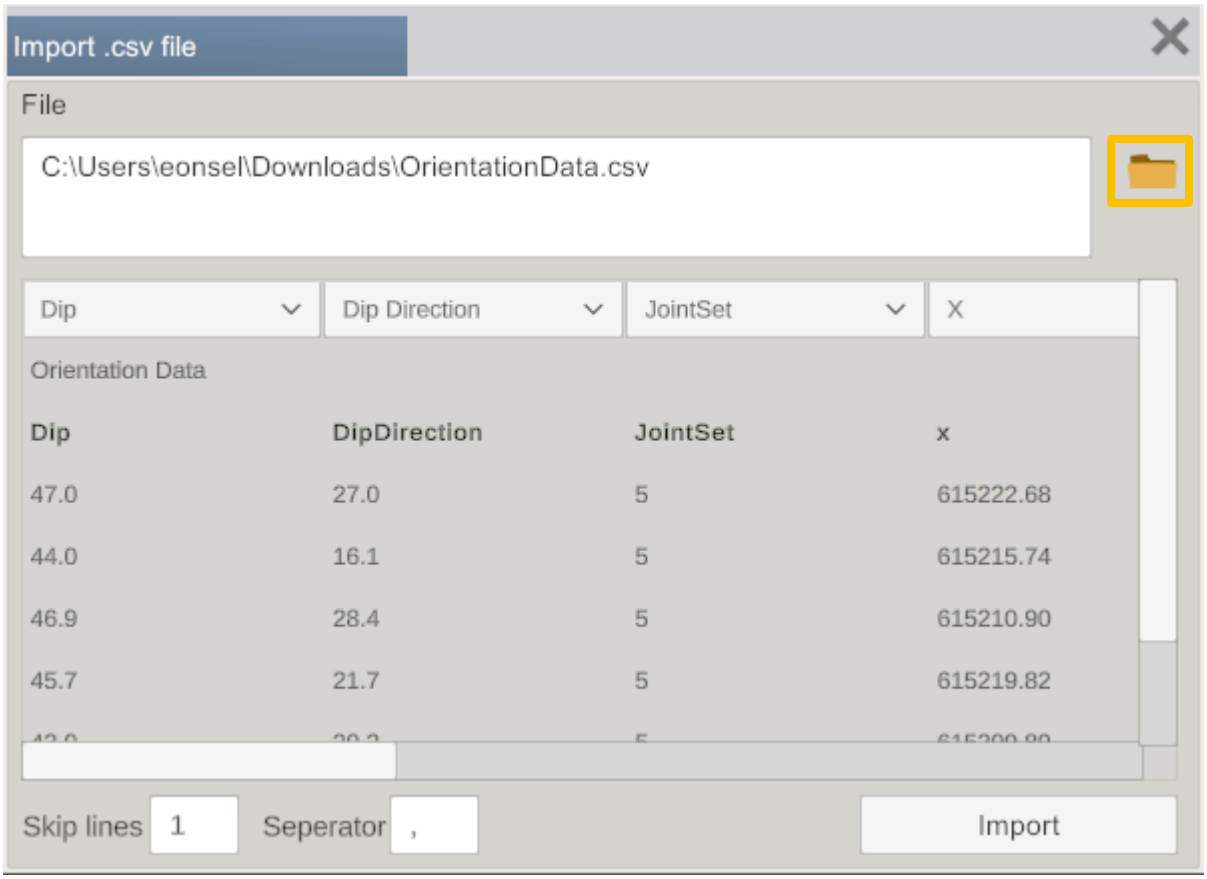


Right click the file and click the Active File. Next to the filename a √ symbol will be displayed to mark the file as active file. The new discs that are created will be saved into this active file.

To import orientation data from a .csv file, right click the Discs branch and click the Import button.

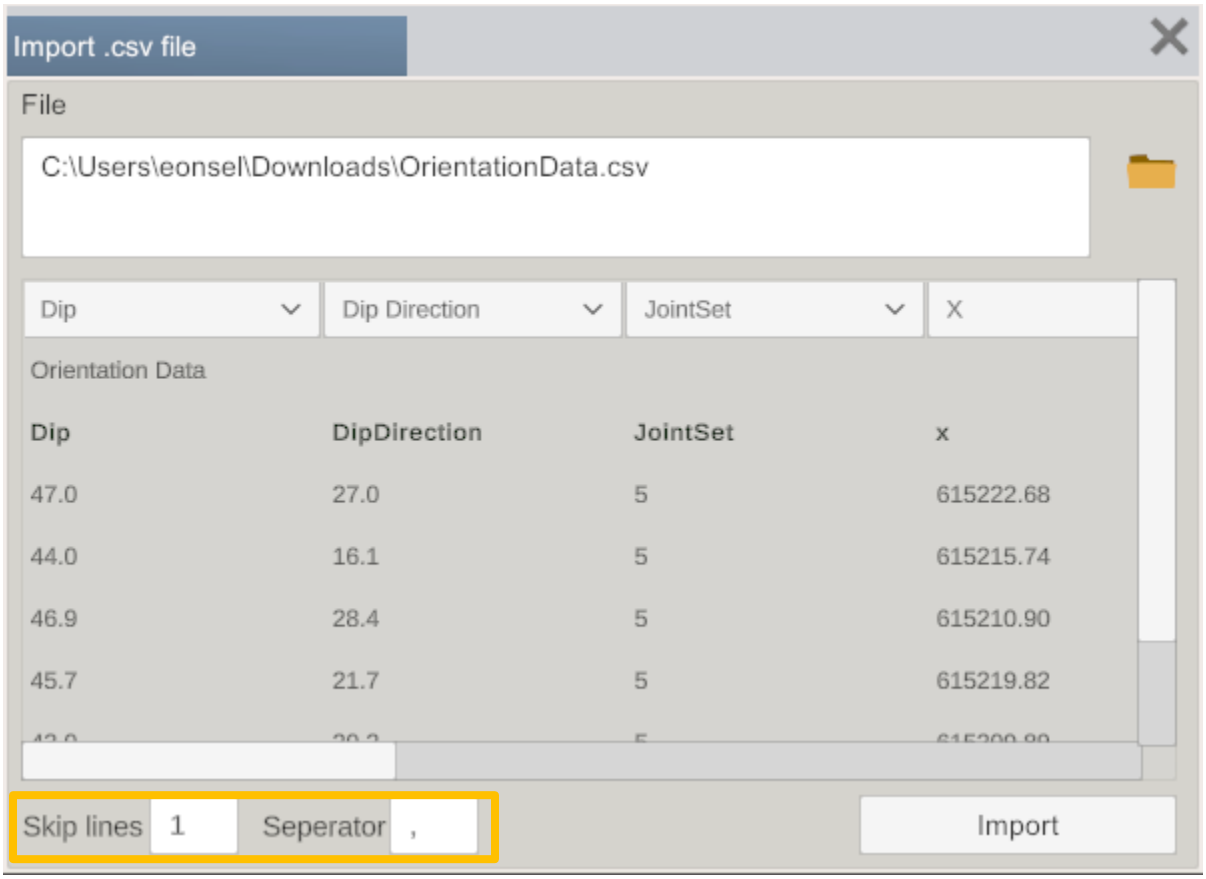


Click the folder button to select the .csv file.



[Table of Contents]

Adjust the “skip lines” and “seperator” parameters to correctly parse the .csv file.



Select the matching columns for dip, dip direction, joint set, x, y, z, diameter, etc. Click import to create the new orientation data file.

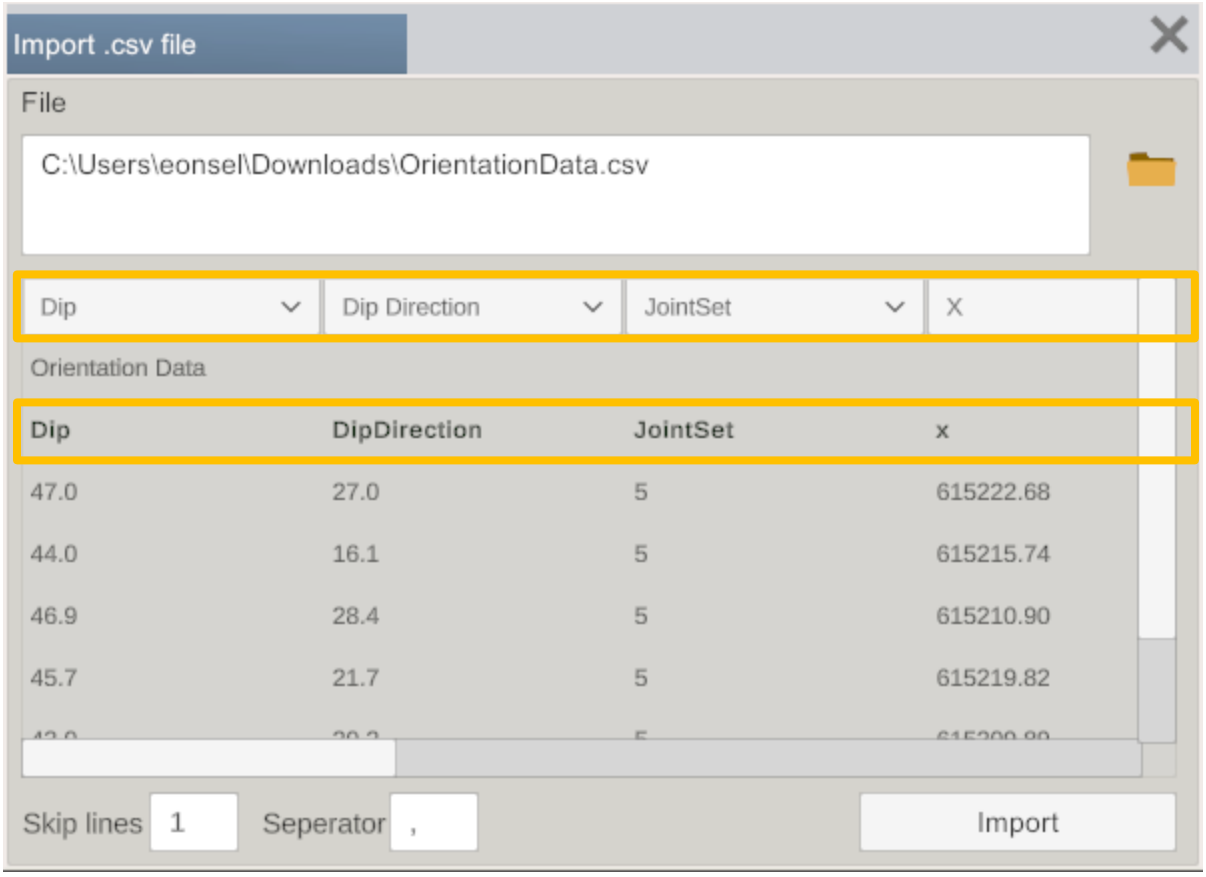
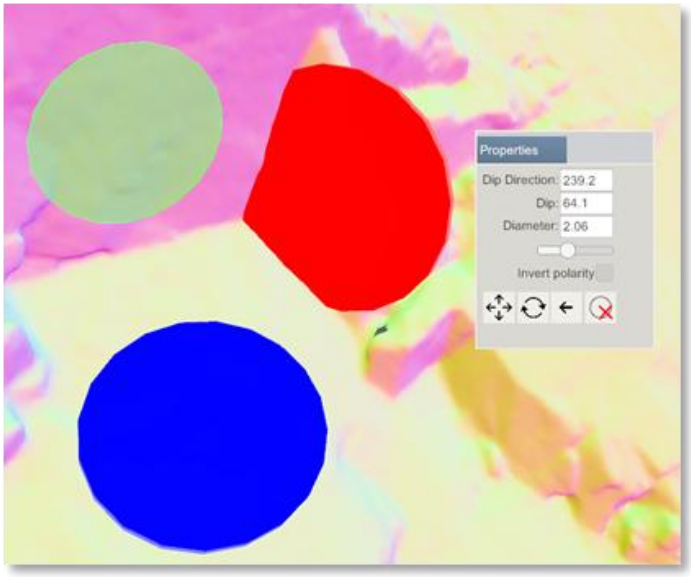


Table of Contents

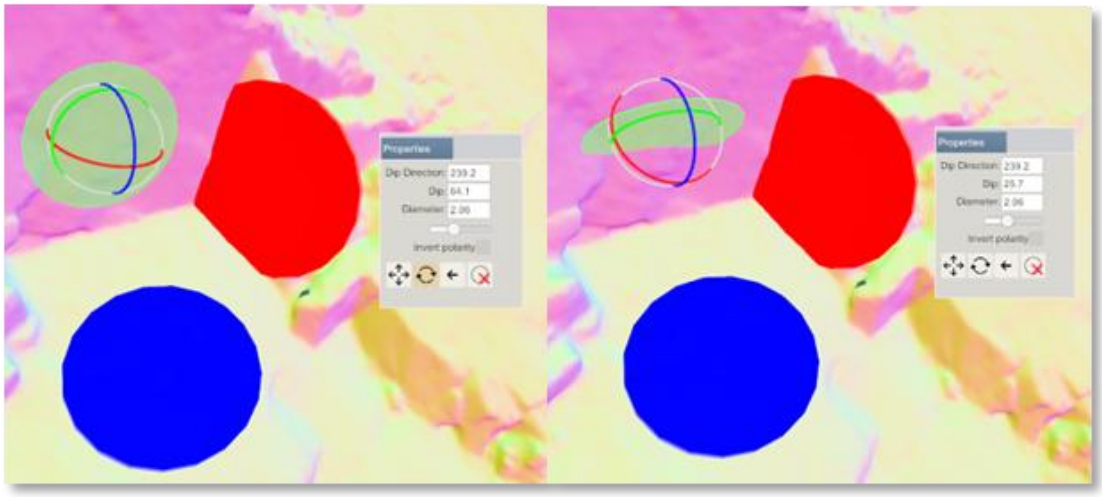
Left click a disc to access the disc properties. Diameter can be modified by editing the diameter textbox or slider.



Orientation can be modified with two methods:

1. Edit the Dip Direction and Dip text boxes or use the mouse scroll after clicking these text boxes.
2. Click the rotate  button. For each axis, 3 circles on the disc will be displayed. Click one of these circles and rotate the disc by dragging the mouse. Click the reset button  to reset the rotation change.

Table of Contents



To change the location of the disc, click the move button  . For each axis and each axis combination, 3 arrows and 3 planes will be displayed on the disc, respectively. Click one of these arrows or planes and move the disc by dragging the mouse. Click the reset button  to reset the location change.

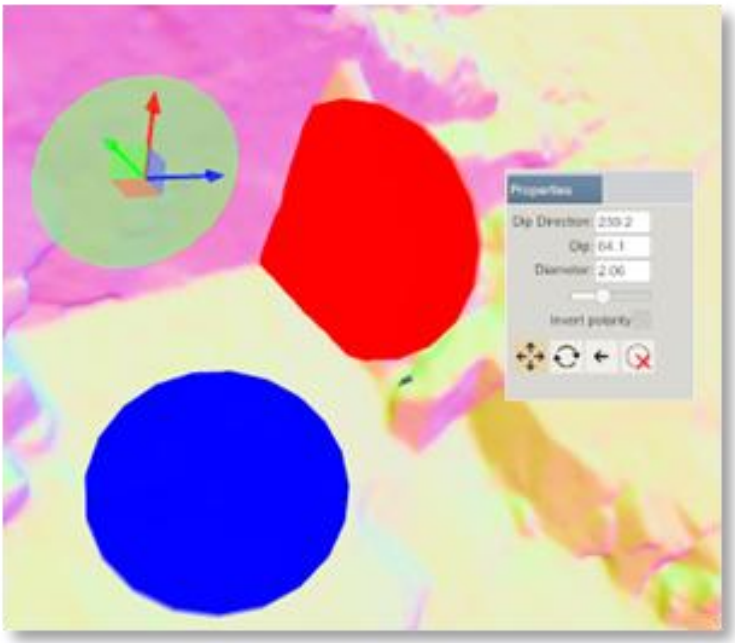
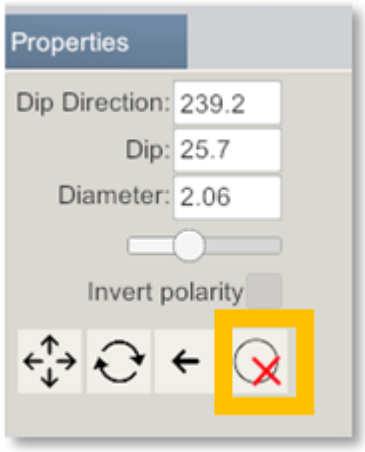


Table of Contents

A disc can be deleted in one of three ways. Select the disc and:

1. Press the Delete key.
2. Click the Delete button on the ribbon menu.
3. Click the Delete button on the Properties panel.

Click the Delete button on the Properties panel.



To remove all of the discs, click the Clear button on the ribbon menu.

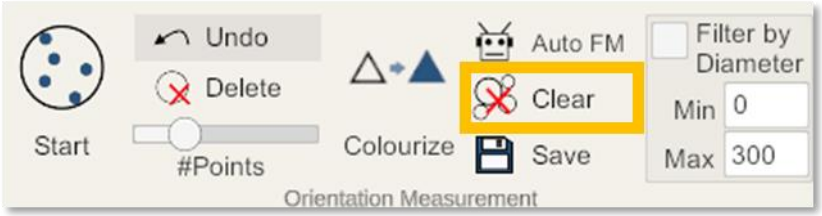
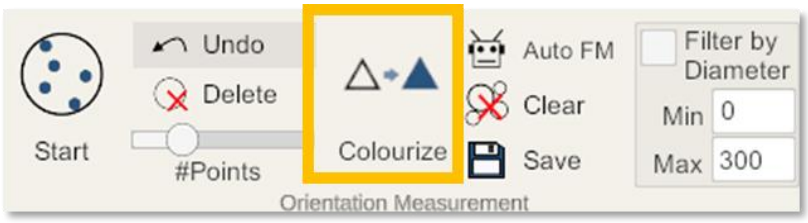
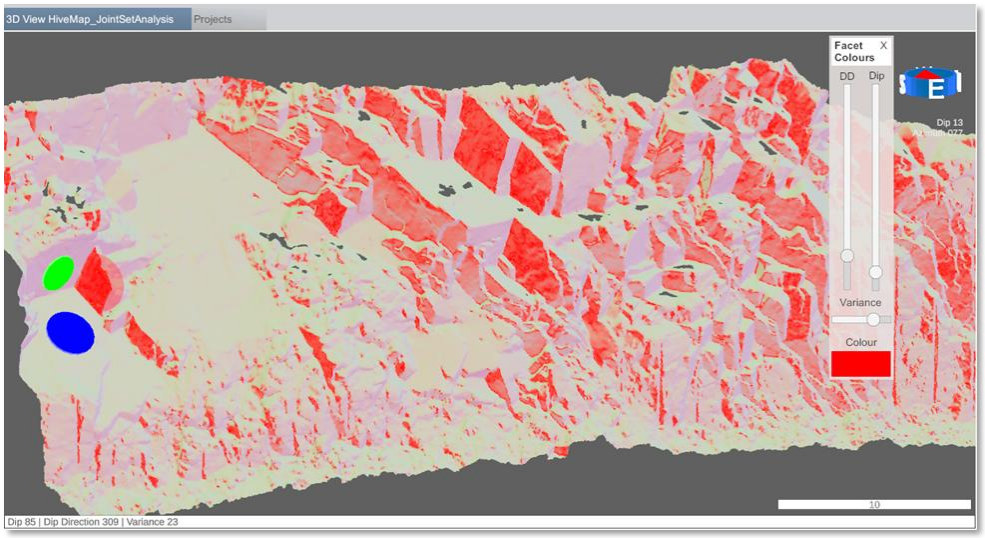


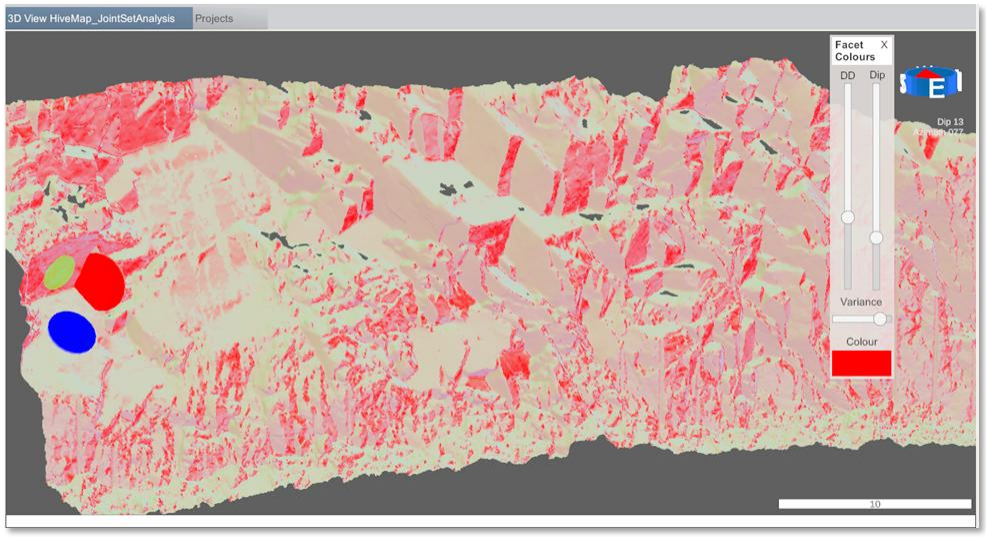
Table of Contents

The Colourize function is used to highlight the mesh triangles with a specific orientation.



Click the "Colourize" button, and a menu with three sliders will appear in the 3D View menu. Use these sliders to set the Dip, Dip Direction, and Variance. The slider values are displayed in the 3D view information bar.
Click the colour button to change the highlighting color.
If you click a disc, the sliders will automatically adjust to match the disc's orientation.





[Table of Contents]

Check "Filter by Diameter" to filter the discs based on their diameters.

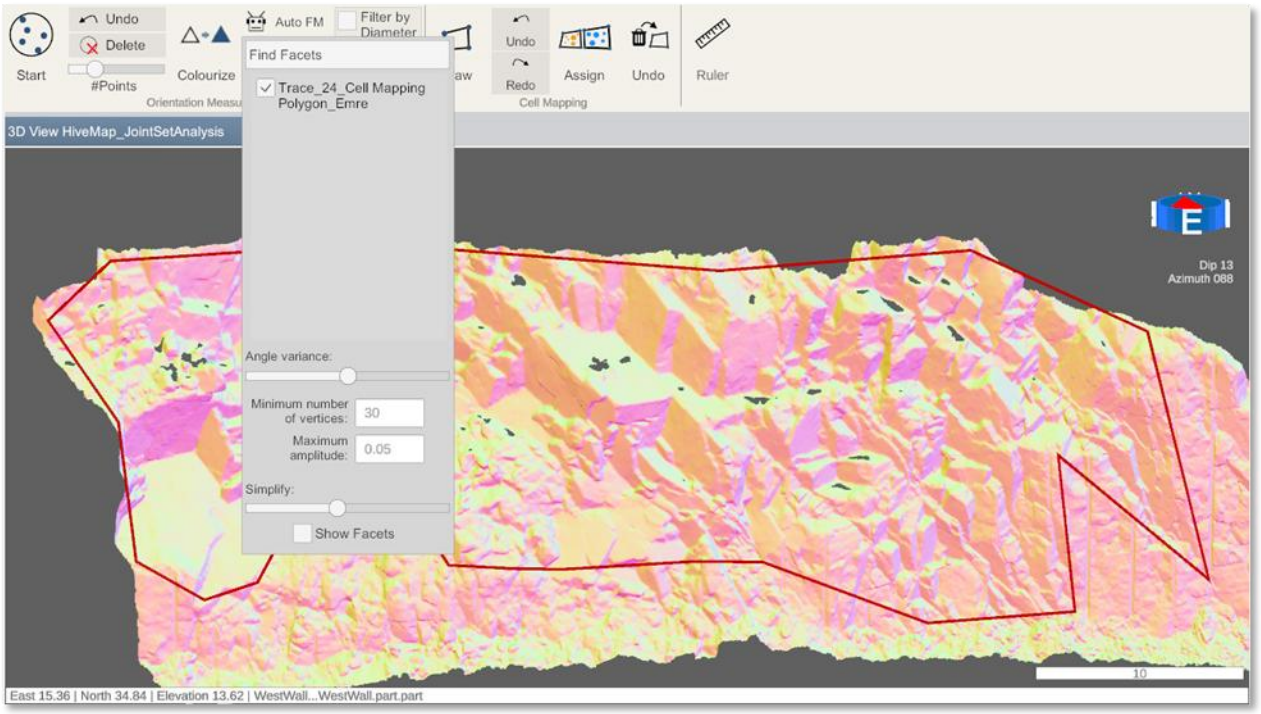
3.4.2.1 Auto Facet Mapping

The Auto Facet Mapping algorithm searches for facets and creates discs for each facet that it detects. This algorithm works with cell polygons (refer to Section 3.4.5).

Click the Auto FM button. A menu will be displayed with these options:

- Cell list: Select the cells where the facets will be searched.
- Angle variance: Each vertex of the mesh has a normal. Vertices from the same facet will be oriented to the same direction with some variability due to facet surface roughness and waviness. Using the variance parameter algorithm, it decides to add or remove vertices to the facets.
- Minimum number of vertices: If a facet has less vertices than this parameter, it will not be included.
- Maximum Amplitude: A vertex will be part of the facet if the distance from the facet plane less than this value.
- Simplify: It is the ratio of vertices that will be used in the algorithm. When the parameter is 1 all the vertices will be used. This will cause the algorithm to run longer.
- Show Facets: After running the algorithm discs will be generated on the facets. if Show Facets is checked, vertices will be coloured. If they are from the same facets, they will have the same colors. Each facet will have a random colour.

Table of Contents



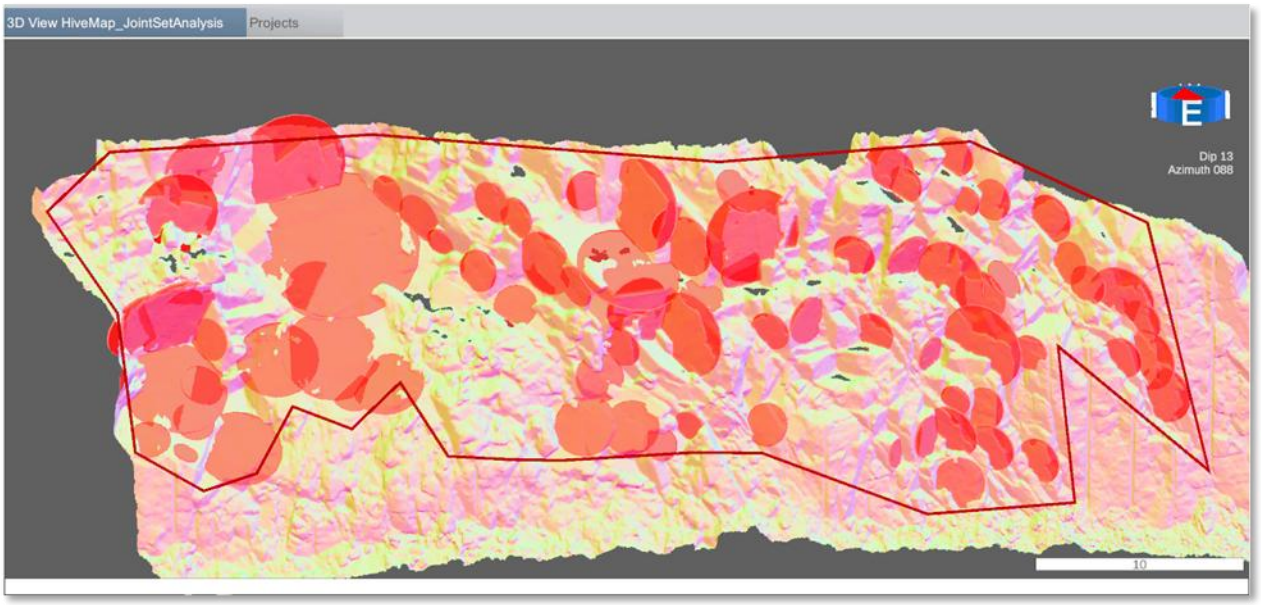
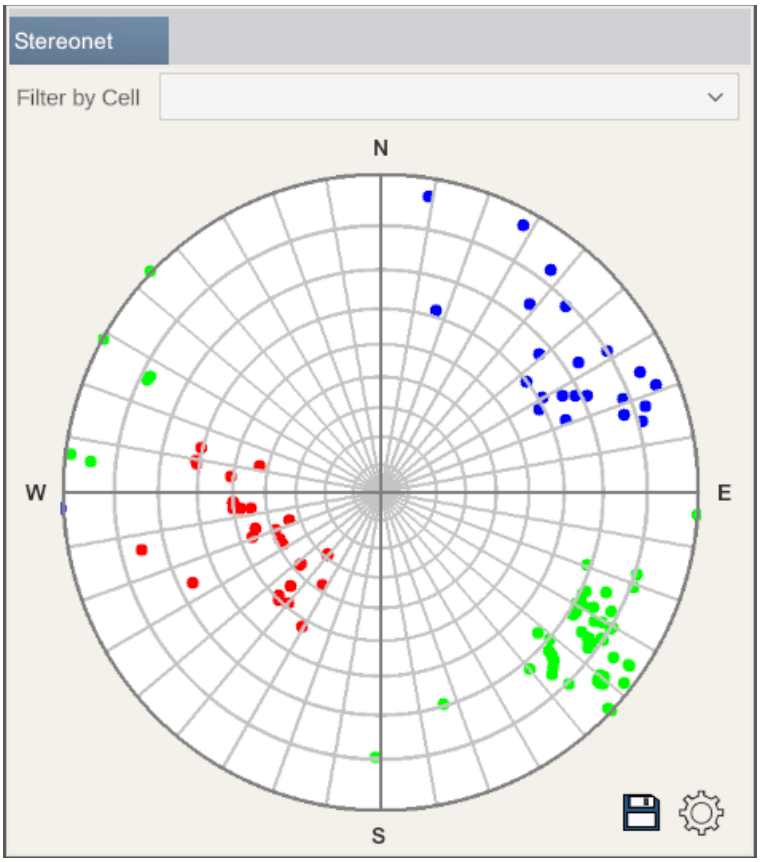


Table of Contents

3.4.3 Stereonet

Poles of discs, cones, joint sets, and kinematic analysis elements such as friction cone, daylight area, etc. are displayed on the Stereonet. The Save and Stereonet Menu button is in the bottom right corner. Save button saves the stereonet to Stereonet.jpg file in the project folder.



Show Poles option displays disc poles as dots and cones as triangles.

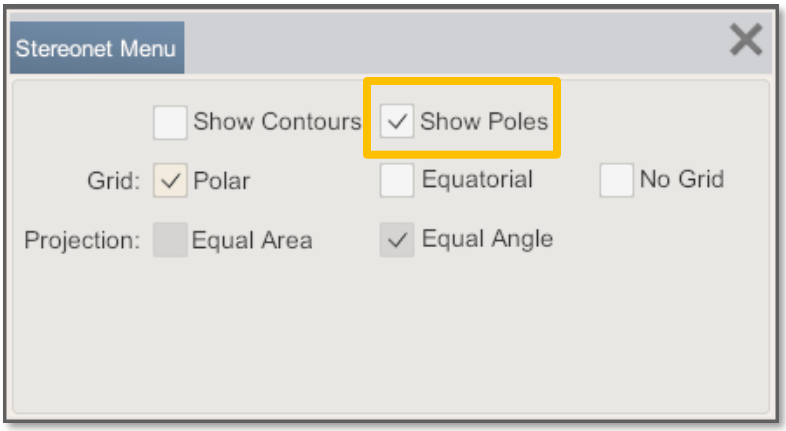
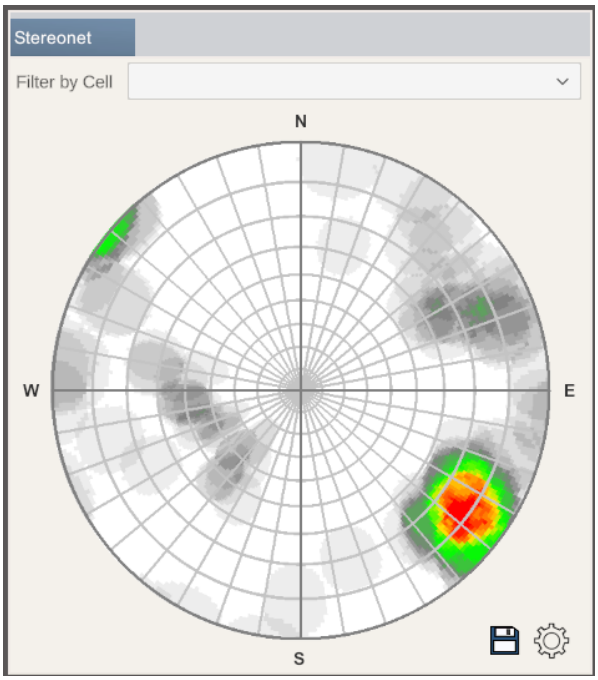


Table of Contents

Show Contours displays the density of disc and cones.



The grid can be removed or selected as polar or equatorial. The projection is set to equal angle, as equal area is not yet supported.

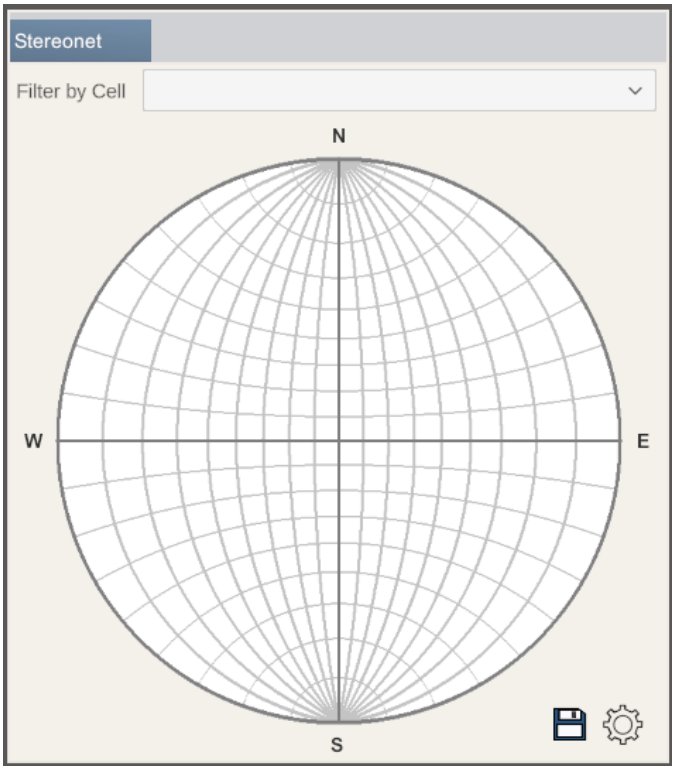
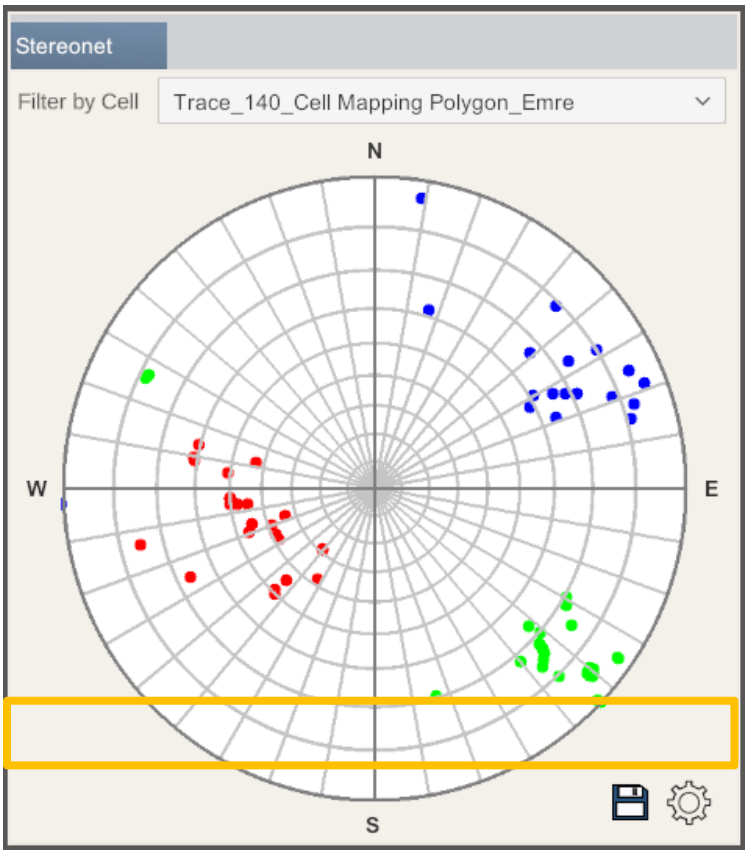


Table of Contents

Data on the Stereonet can be filtered using cells (Section 3.4.5). If no cell is selected, all the data will be shown.

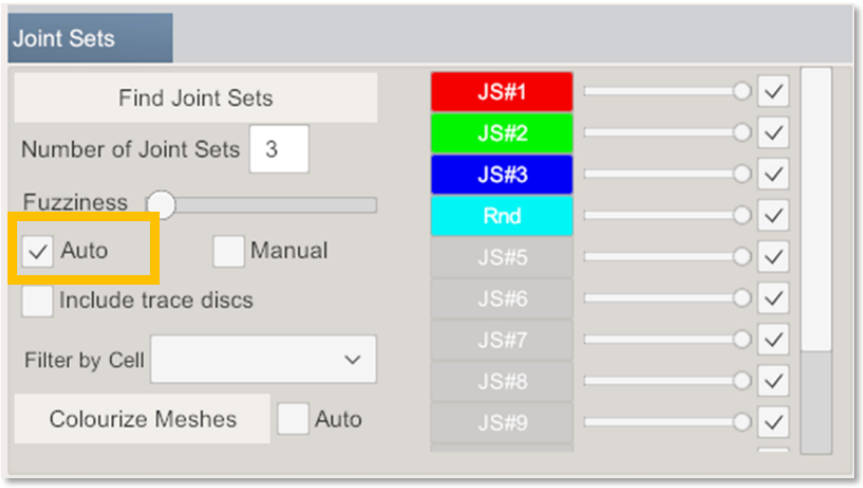


3.4.4 Joint Sets

3.4.4.1 Clustering

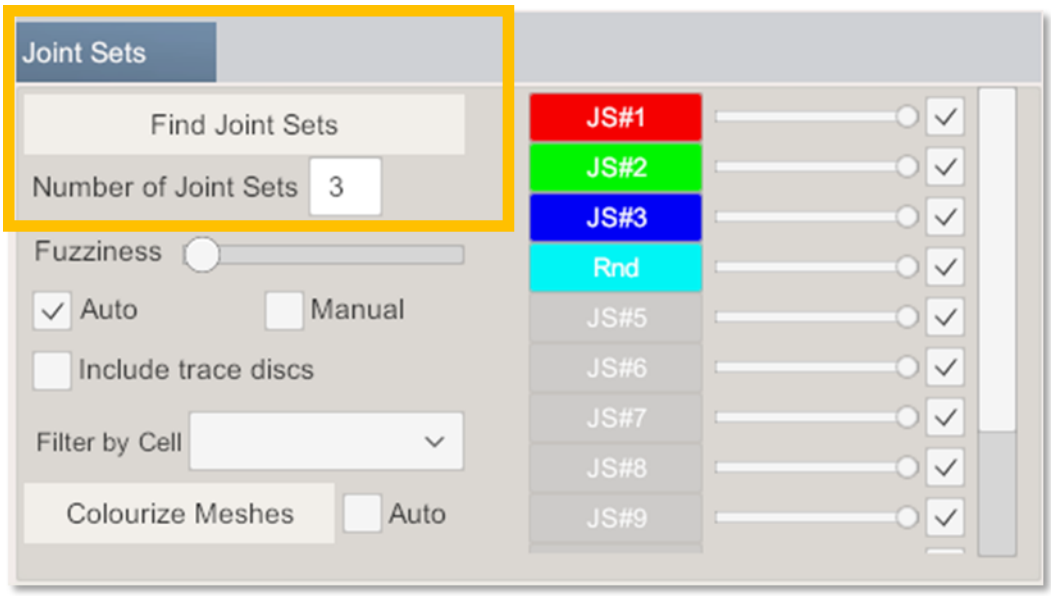
In the Joint Sets panel, joint sets can be determined using a Fuzzy K-Mean algorithm, either automatically or manually.

To find the joint sets automatically, check the Auto box.



Auto method algorithm requires 2 parameters:

1. Number of joints: This should be estimated by using the stereonet and interpreting the rock face.
2. Fuzziness: If fuzziness is 0, the algorithm will assign every joint to a joint set. If there are random joints which are not close to any joint set center, they should not be assigned to a joint set. This can be adjusted with increasing the fuzziness parameter. The higher the fuzziness is, the less dispersed the joint sets will become.



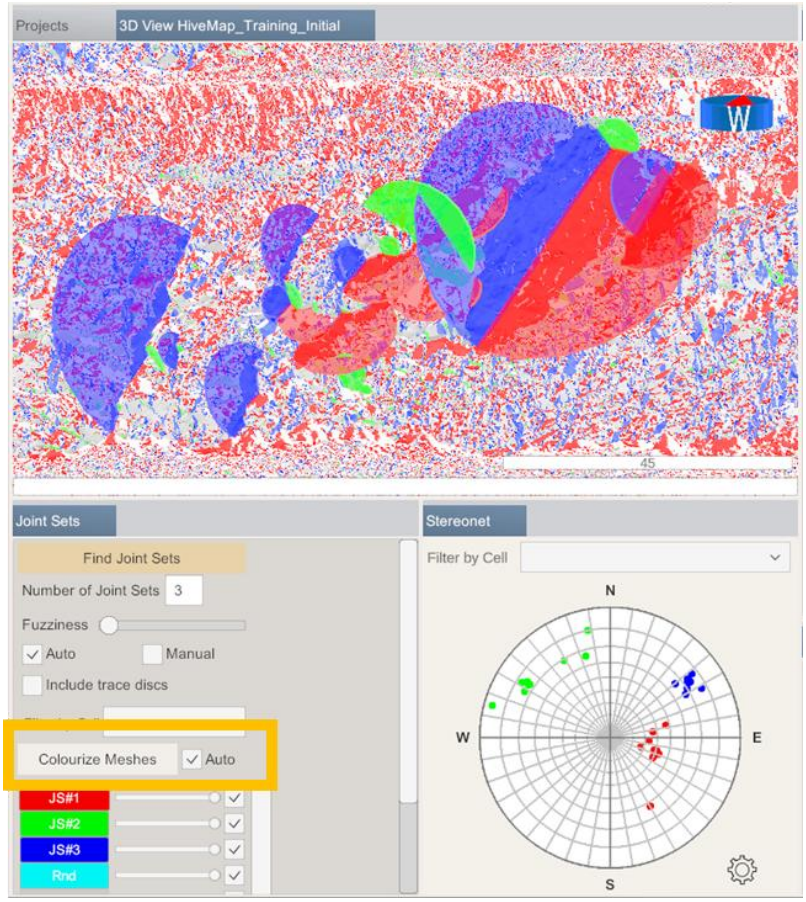
After running the algorithm, joints are separated into sets and each joint pole on the stereonet, and the disc on the 3D view, are colourized into their joint set colour.

If the Colourize Meshes button is clicked or Auto box next to it is checked, the triangles, which have the same orientation as the joint sets, are colourized too.

Table of Contents

Joint sets are listed in the Joint Sets list.
Each joint set has a slider, a check box, and a button in the joint set colour.

- The slider changes the opacity of the joint set discs.
- The checkbox hides and unhides the discs of the joint set.
- The button opens the joint set menu.



To determine the joint sets manually, check the Manual box. This will disable the Fuzziness slider and the button text will change to “Assign Joints to Joint Sets”.

Table of Contents

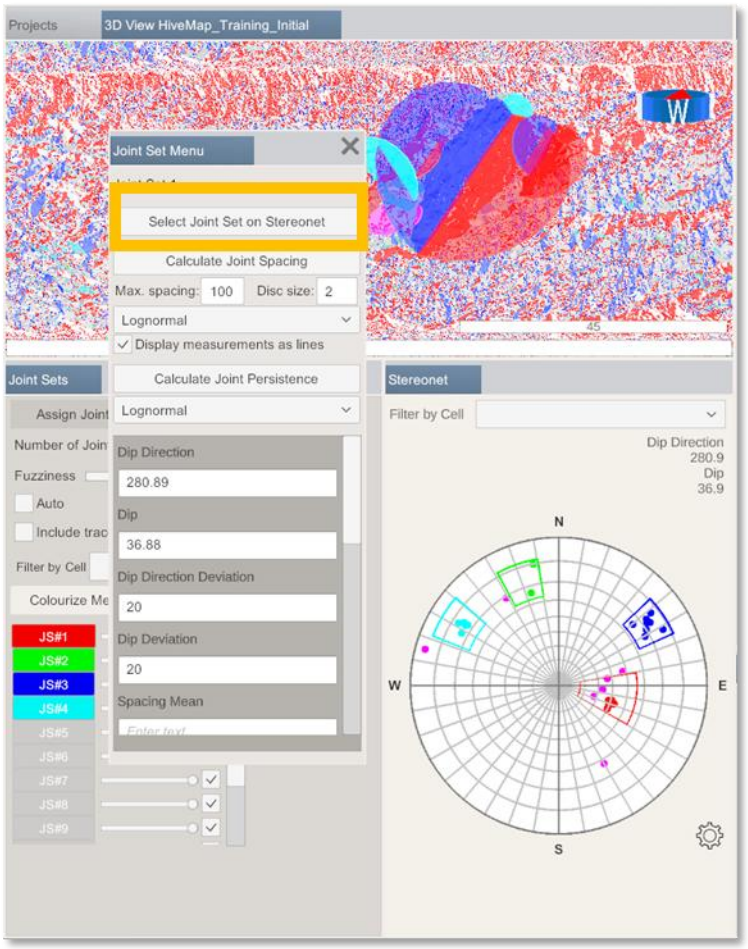
Enter the number of joint sets.

Click the Assign Joints to Joint Sets button.

On the joint set list, click the JS#1 button to open Joint Set 1 menu.

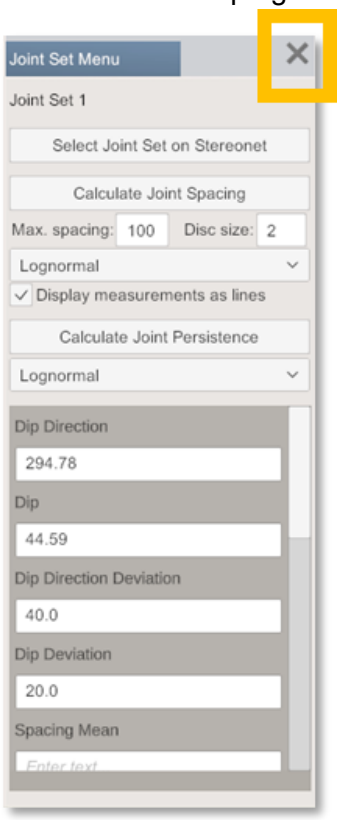
Table of Contents

Click to select the Joint Set on the Stereonet. Adjust the Dip Direction Deviation and Dip Deviation values in the form. Then, click on the Stereonet and drag the mouse to reposition the Joint Set 1 window on the Stereonet.



[Table of Contents]

Click “X” on the top right corner of the menu and click the Save changes and close button.



Click Assign Joints to Joint Sets button. All the joints inside Joint Set 1 window will be assigned to the Joint Set 1.

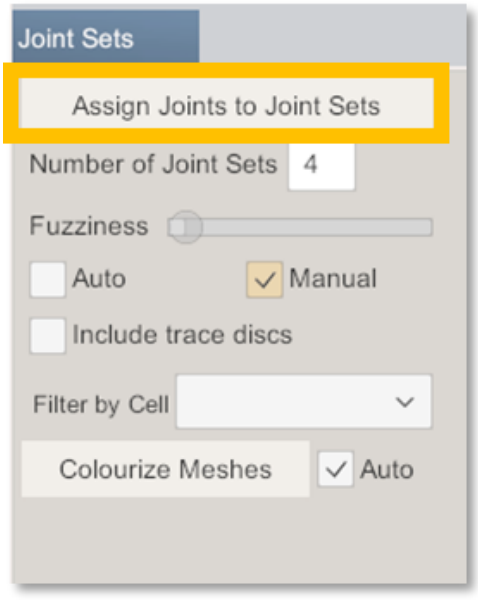
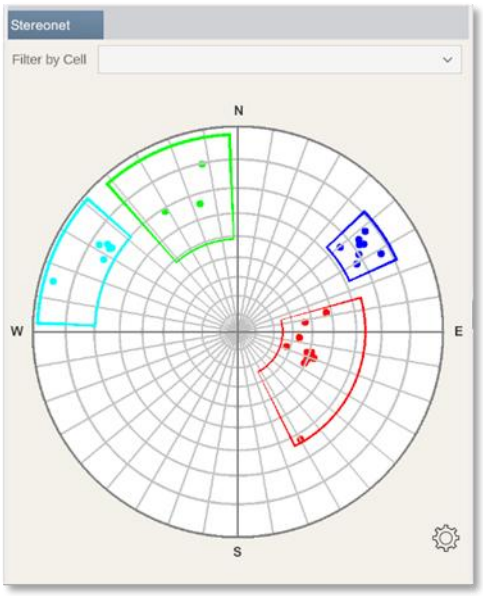


Table of Contents

Repeat this process for the other joint sets.



3.4.4.2 Joint Set Spacing

To calculate joint set spacing for a joint set open the joint set menu of that joint set.

The Joint Spacing algorithm measures the distance between each disc pair, and if they intersect when they are projected in the joint set orientation.

- Display measurements as lines: This will display green or red lines in the disc center. If there is a measurement, line will be green. Users can validate the algorithm by checking the measurements.

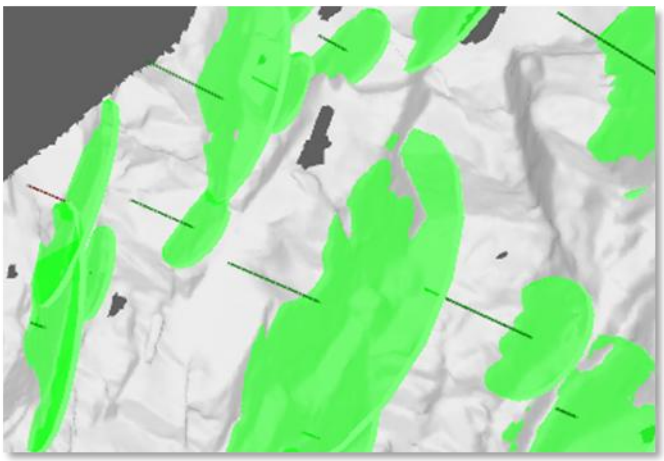
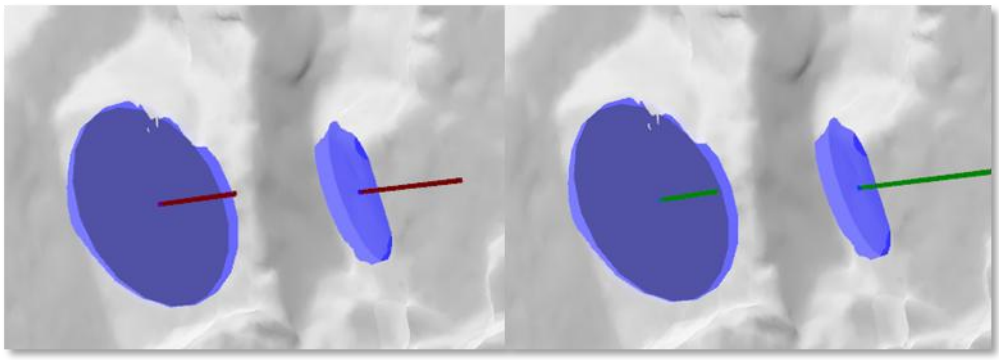


Table of Contents

- Max. spacing: If the distance between two discs is larger than the maximum spacing value, it will be ignored.
- Disc size: This is the disc diameter scale during the calculation. If the disc size parameter is 1, the disc diameters stay same during the calculation. 2 will double the disc diameters. If the discs are mapped smaller than they should be, the disc diameters can be increased to ensure that there is measurement between these discs.
- Distribution: Lognormal or Negative Exponential can be selected. This will be displayed on the histogram.



Click the Calculate Joint Spacing button to run the algorithm.

A histogram will be created with the selected distribution.

Mean, standard deviation, min, max values will be calculated. Raw data are saved in the JointSets folder in the Project folder as .csv file.

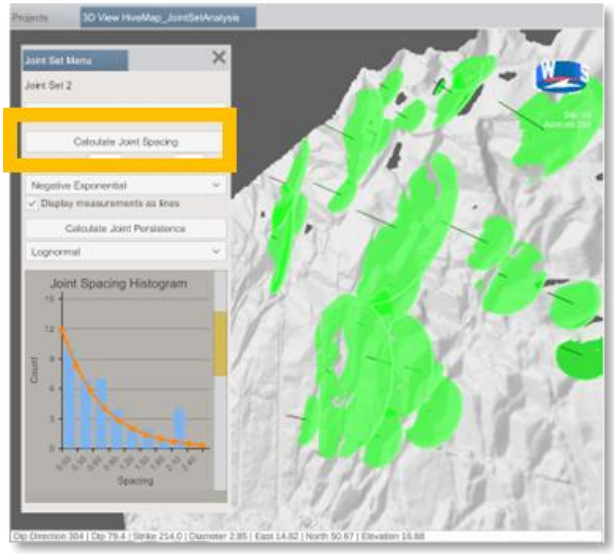
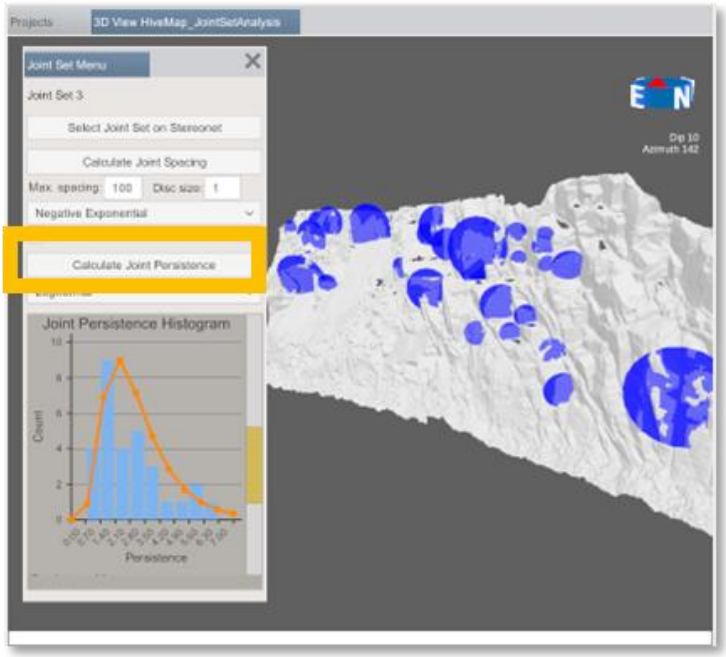


Table of Contents

3.4.4.3 Joint Set Persistence

Click Calculate Joint Persistence to calculate disc size statistics of the joint set.

A histogram will be created with the selected distribution.

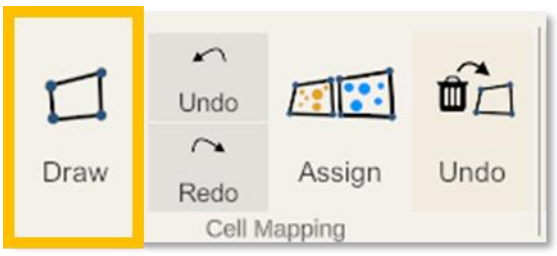


3.4.5 Cell Mapping

With the Cell Mapping tool users can draw polygons which can be used to:

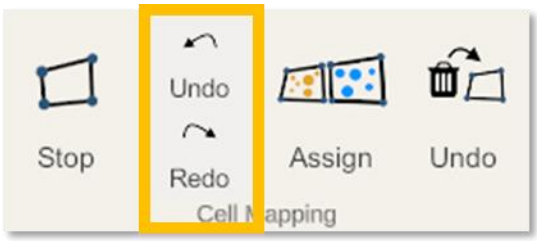
1. Do geotechnical cell mapping.
2. Separate the geological domains.
3. Split rock faces according to slope direction for kinematic analysis.

To create a cell polygon, select Draw.

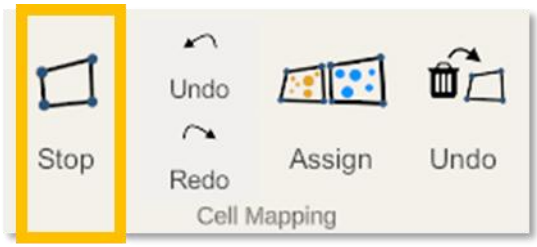


Use your left mouse and do single clicks to draw polygon.

Clicking Undo button or backspace key will delete the last node. Redo button brings back the removed nodes.

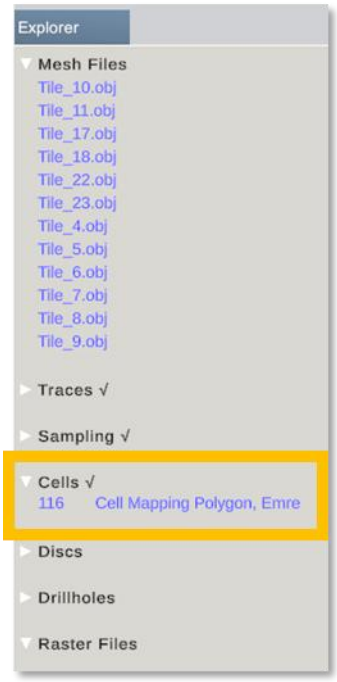


Click the Stop button to finish drawing.

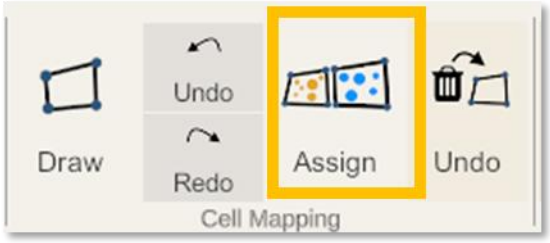


The new cell file is displayed on the Explorer panel under the Cells branch. Cell files are saved to the Cells folder inside project folder.

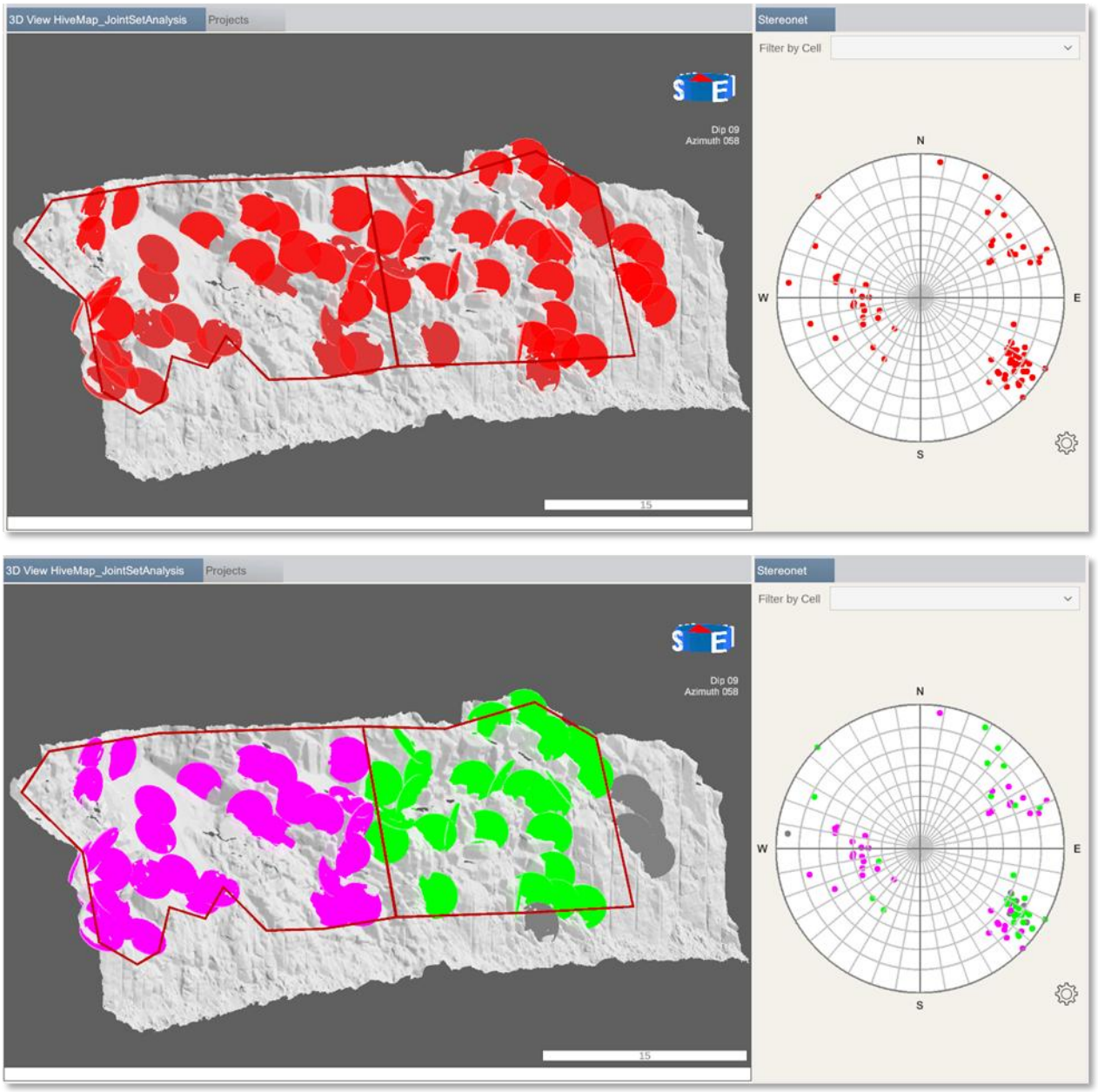
[Table of Contents]



Assign button assigns joints to the cells.

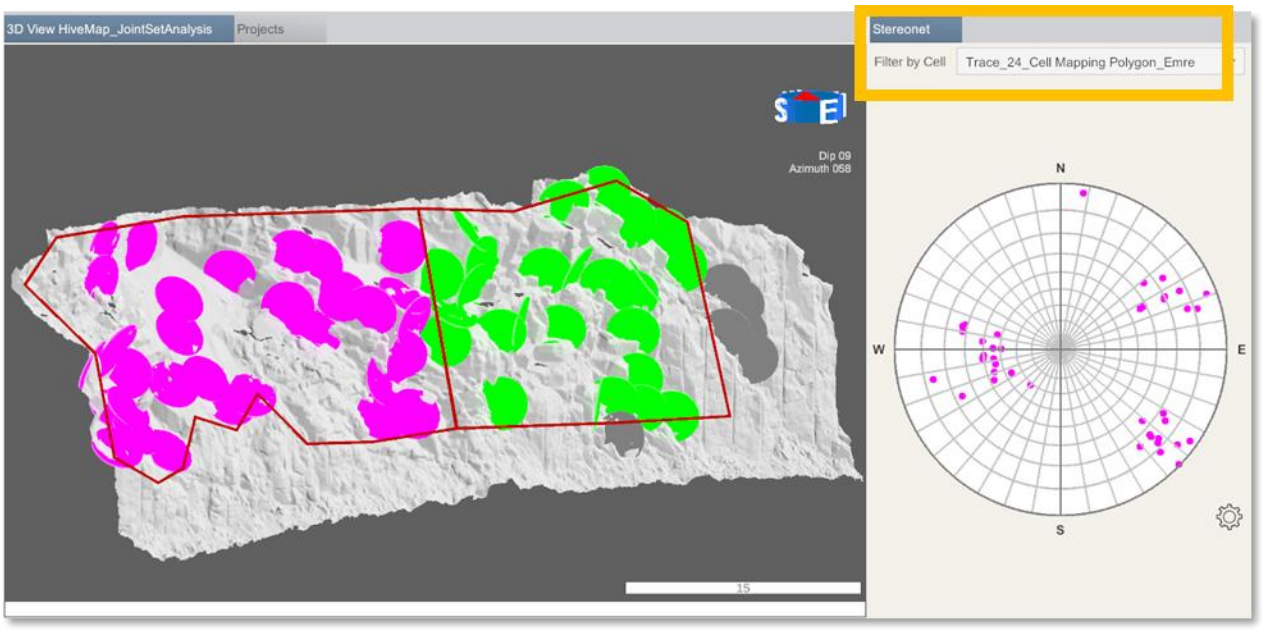


Discs from the same cells are colored identically, while discs that are not in any cell are displayed in gray.



Cells are also listed on the stereonet, joint set menu and kinematic analysis menu. Discs can be filtered by selecting a cell in the Filter by Cell drop down menu.

Table of Contents



The "Undo" button restores the most recently deleted cell. Deleted cells are stored in the Recycle Bin folder within the project folder.

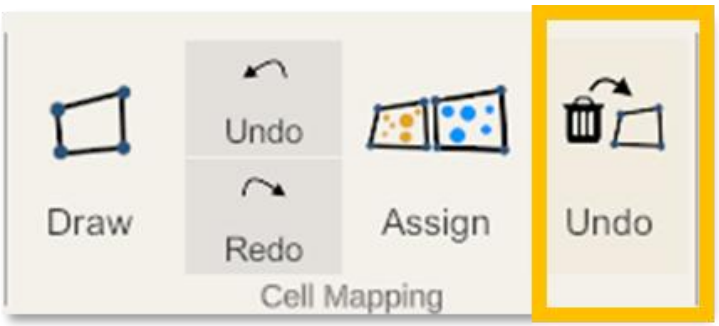
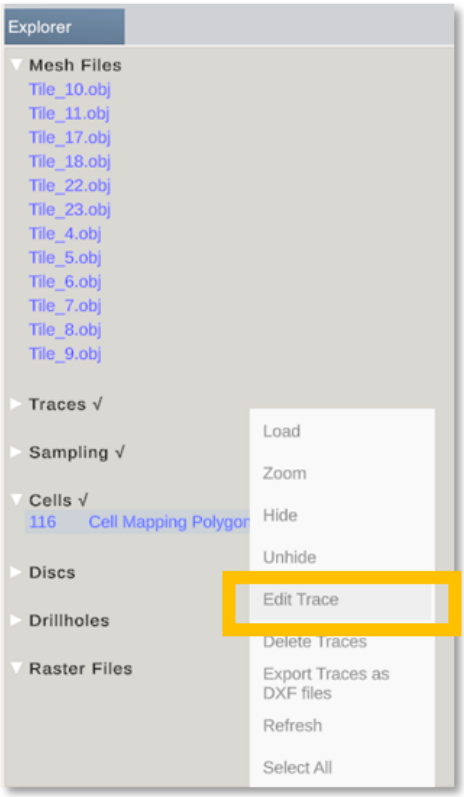


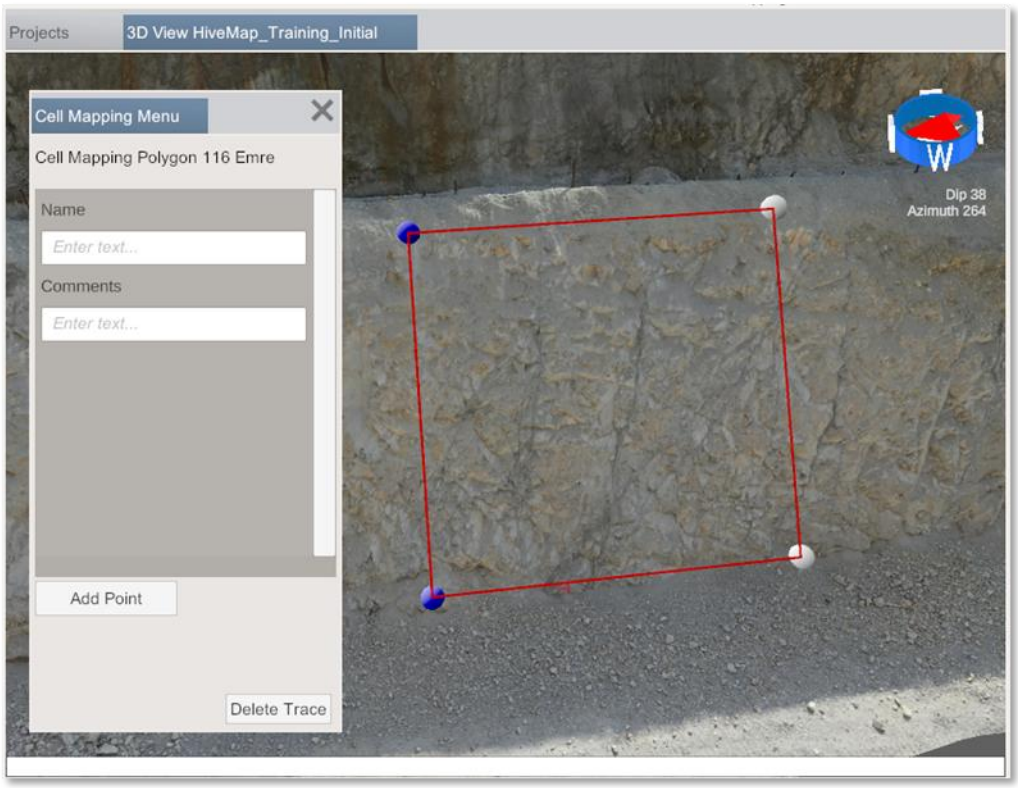
Table of Contents

To edit a polygon, left click the cell in the 3D View menu. Alternatively, right click the cell on the Explorer panel and then the Edit Trace on the Right Click menu.

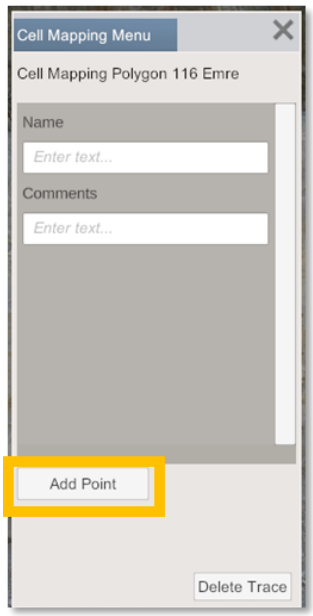


[Table of Contents]

When the Cell Mapping Menu is displayed, and the nodes of the polygon becomes visible.

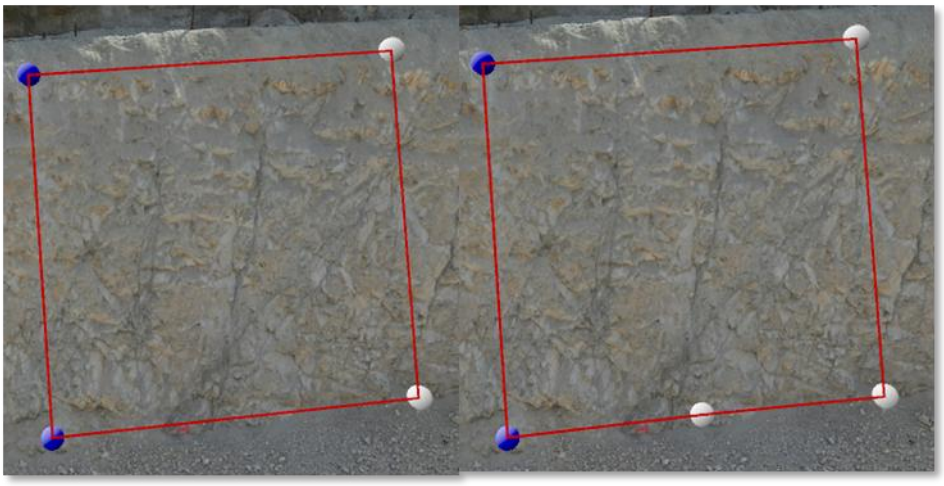


Click the Add Point button and then click the polygon.



[Table of Contents]

A new node will be added where the polygon is clicked.



Click one of the white nodes. The node will become red, and, on the menu, the Delete Point button will become visible. Click this button to delete the selected red node.

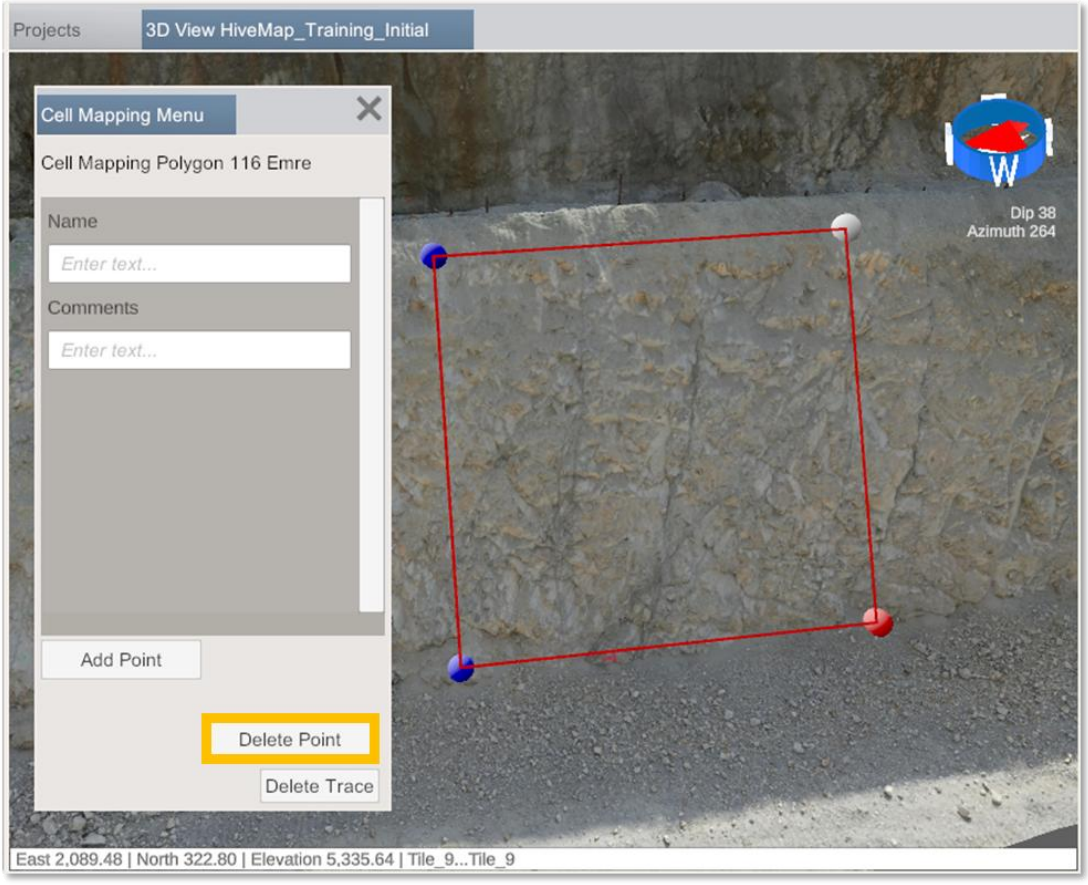
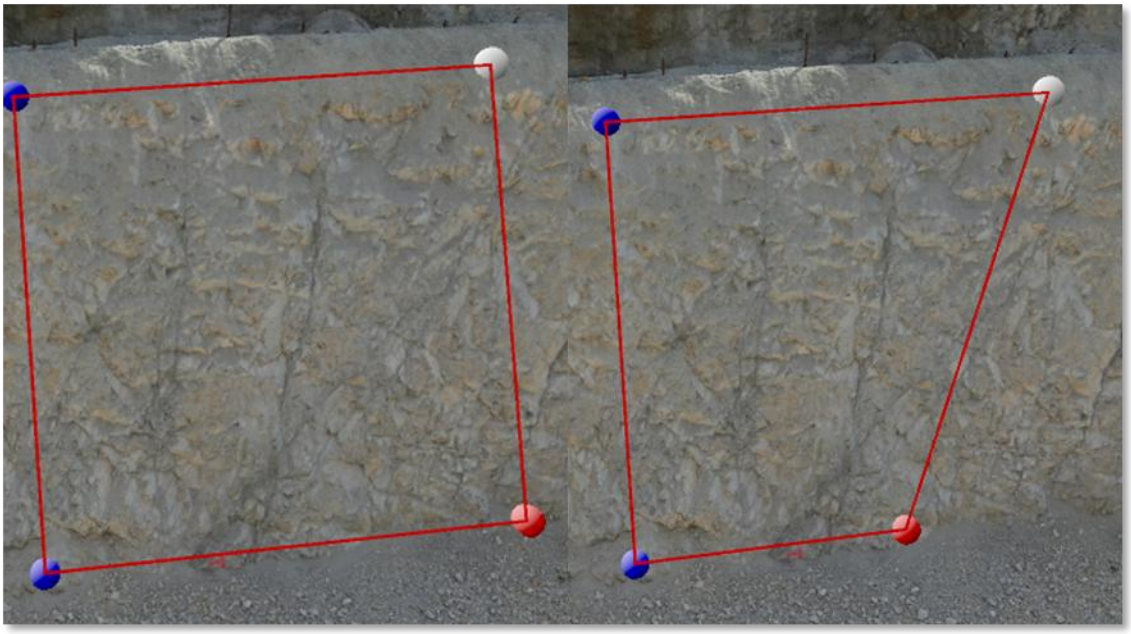
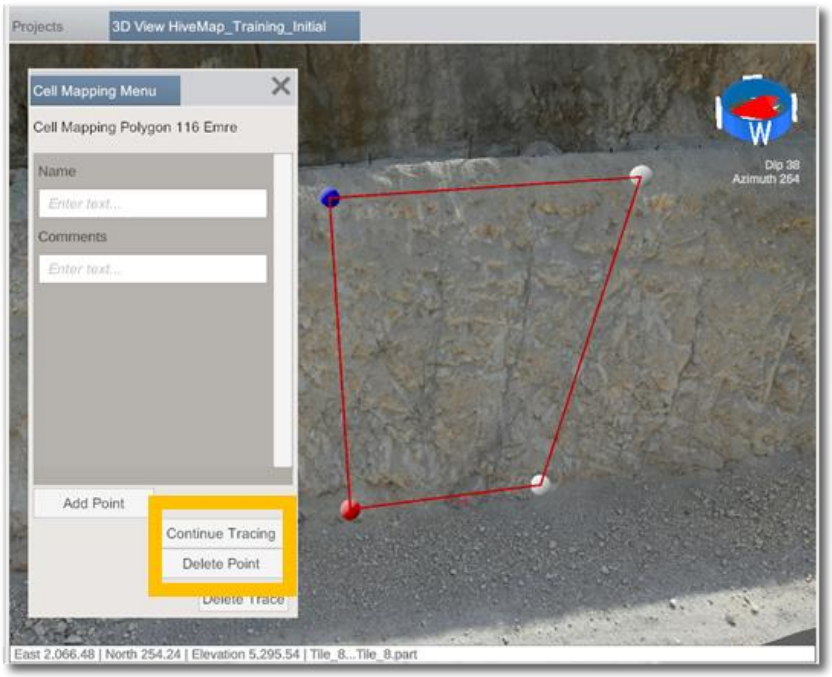


Table of Contents

After selecting node, click the mesh and the red node will move to the clicked position on the mesh.



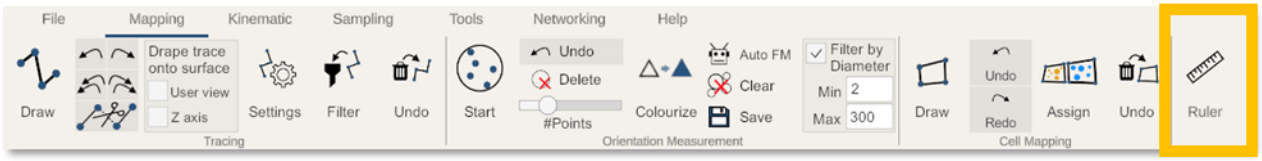
Click the first or last node, which will be purple. The node will turn red, and the "Delete Point" and "Continue Tracing" buttons will appear in the menu. Click "Continue Tracing" to close the trace menu and hide the nodes, allowing tracing to resume from the selected node.



[Table of Contents]

3.4.6 Ruler Tool

The Ruler tool can be used to measure length, vertical length, horizontal length, trend, and plunge between two points on the mesh.



When this tool is activated, a line will be displayed on the screen. The endpoint that is clicked becomes active.

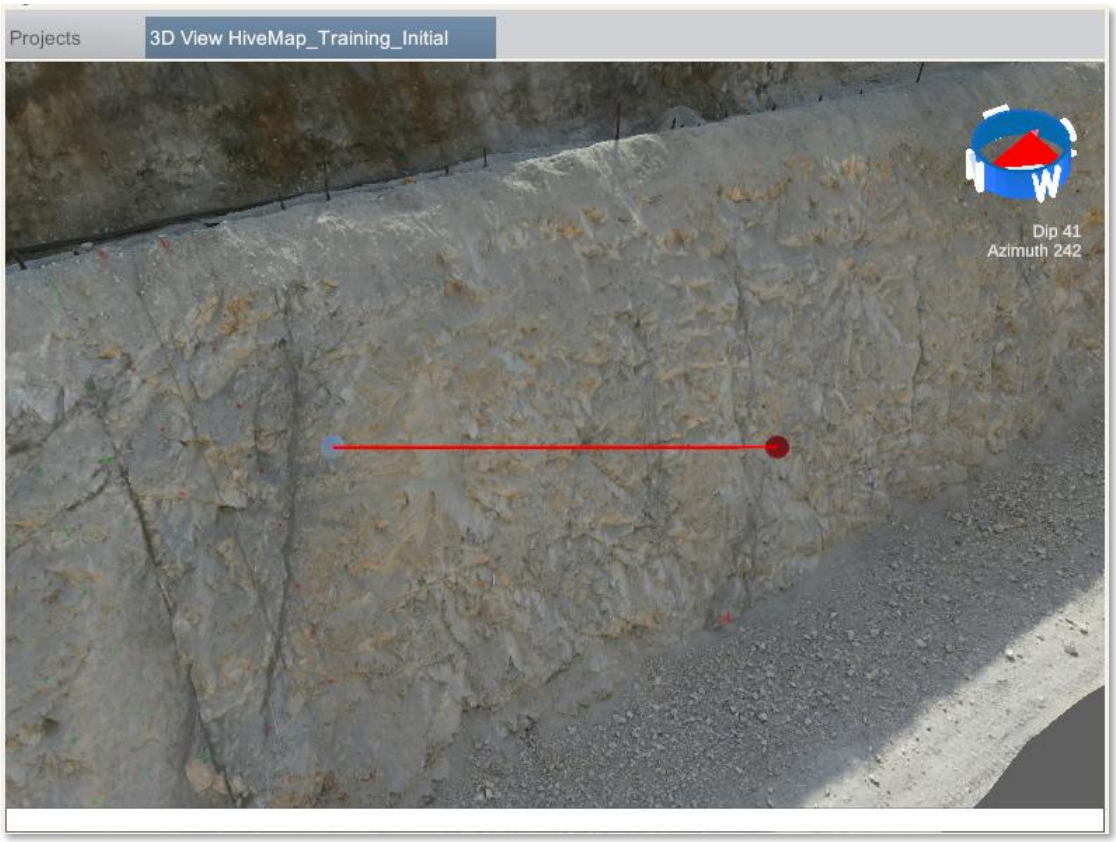


Table of Contents

When the user clicks a point on the mesh the active endpoint will be placed to that point. Measurements will be displayed in the Console panel.

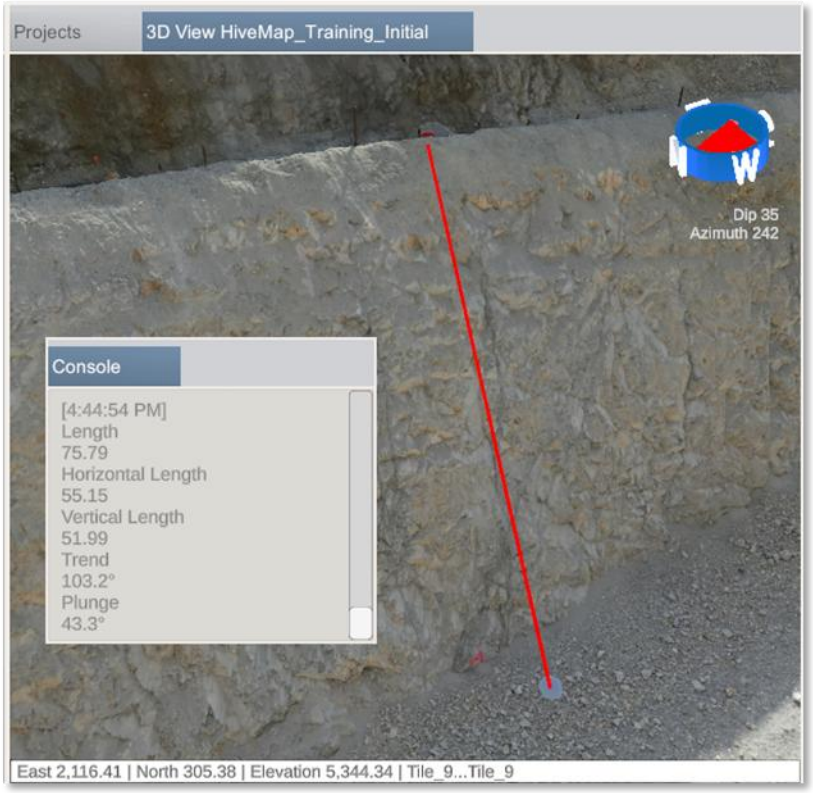


Table of Contents

3.5 Kinematic Analysis

After completing mapping - planar sliding, wedge sliding, flexural toppling, and direct toppling kinematic analysis can be conducted.

Analysis can be done for all the mapping data or data can be filtered using the Filter by Cell tool.

Wedge sliding and direct toppling might happen when there are wedge shaped blocks. Wedge shaped blocks are created by the intersection of joints. To find the intersections between joints click Find Intersection button.

Cones will be created along the disc intersections.

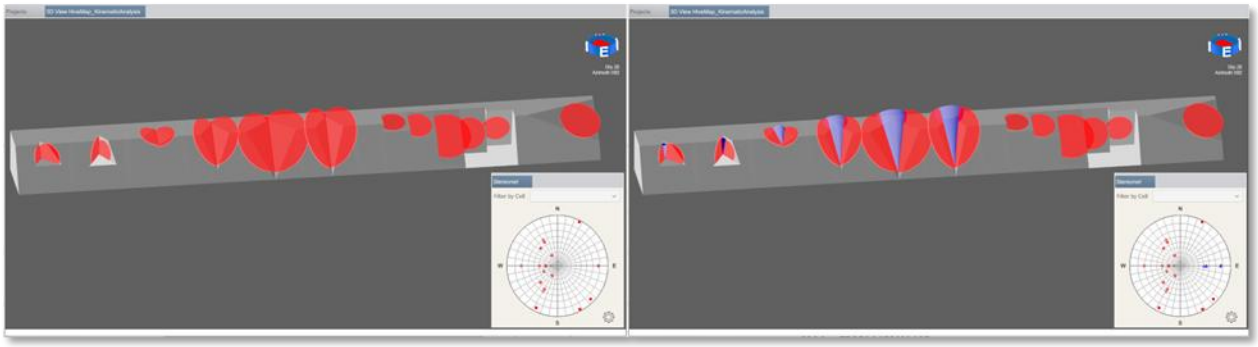


Table of Contents

Remove intersections button deletes the intersection cones.

Slope plane, friction cones, daylight envelope and lateral limits can be displayed on the stereonet.

Planar sliding: For a joint to cause a planar failure, it must be within the daylight envelope, have a dip angle greater than the friction angle (outside the friction cone), and be within the lateral limit. This area on the stereonet is displayed in red. Discs within this area are colored red, while the remaining discs are colored green.

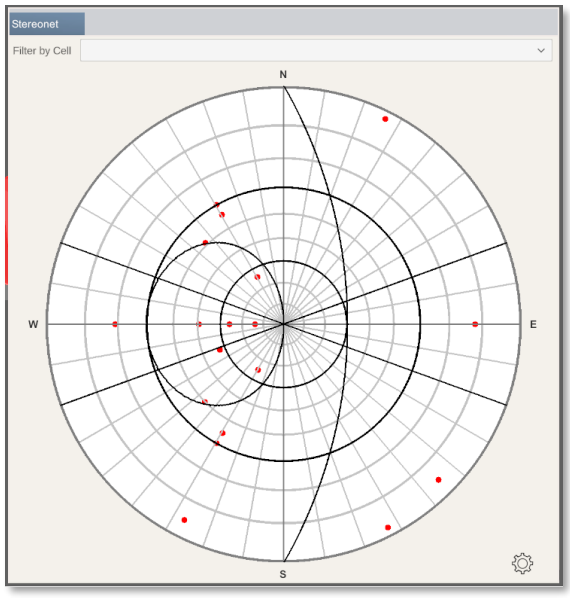
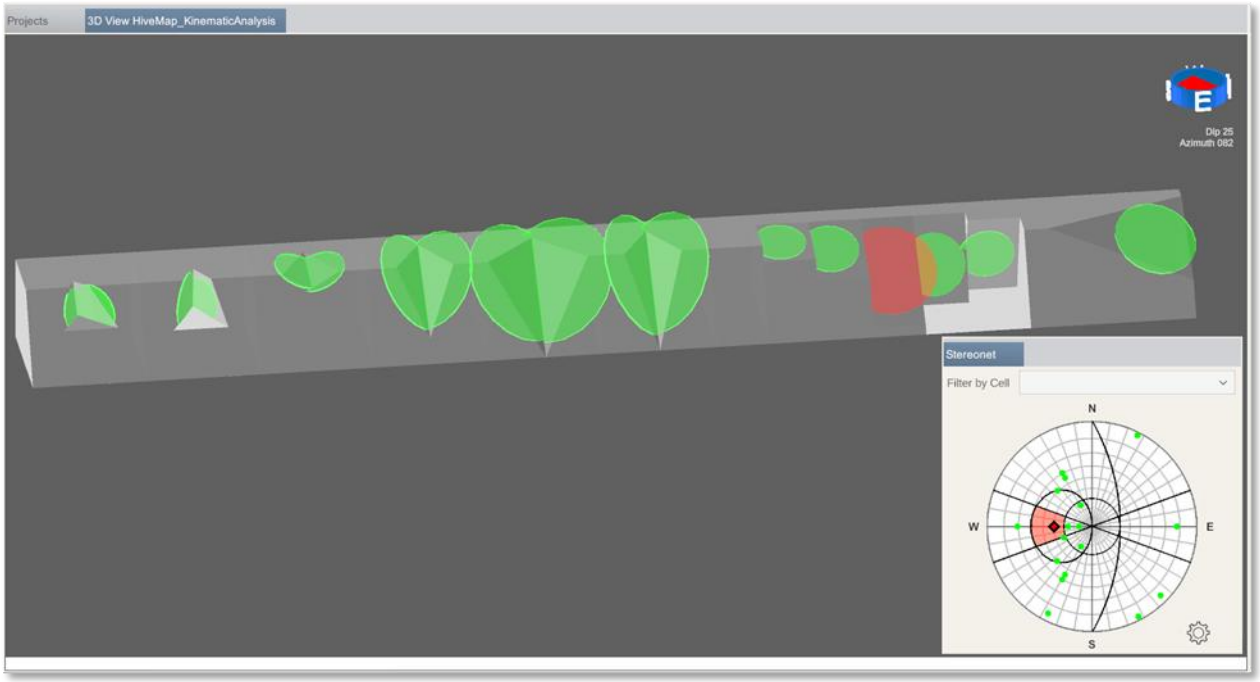


Table of Contents

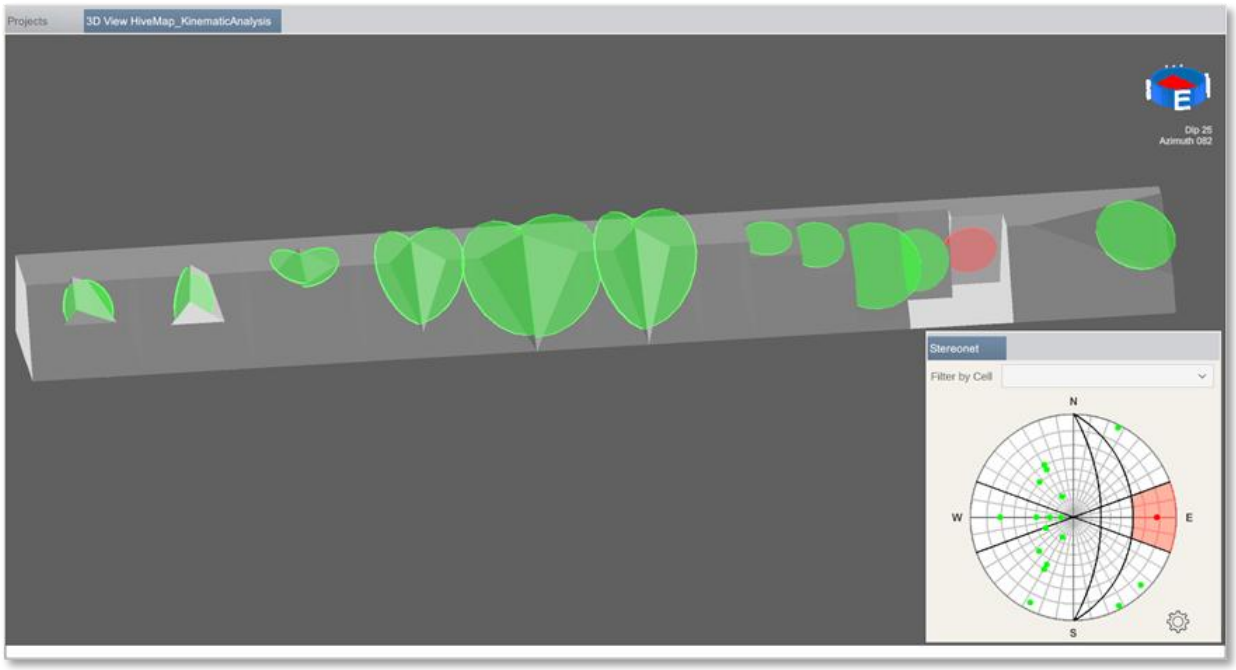


Wedge sliding: Once the disc intersections are calculated, this analysis can be run. If a joint intersection is going to cause a wedge slide, it should be inside the planar friction cone and it should be outside of the slope plane. This area on stereonet displayed with red colour. Cones which are inside this area are coloured in red.

There is a secondary area where the wedge blocks slide on one of their sides. This area is outside of the slope plane, outside of the planar friction cone and inside of the planar friction plane. This area on stereonet displayed with yellow colour. Cones which are inside this area are coloured yellow.

The rest of the cones are coloured green.

Table of Contents



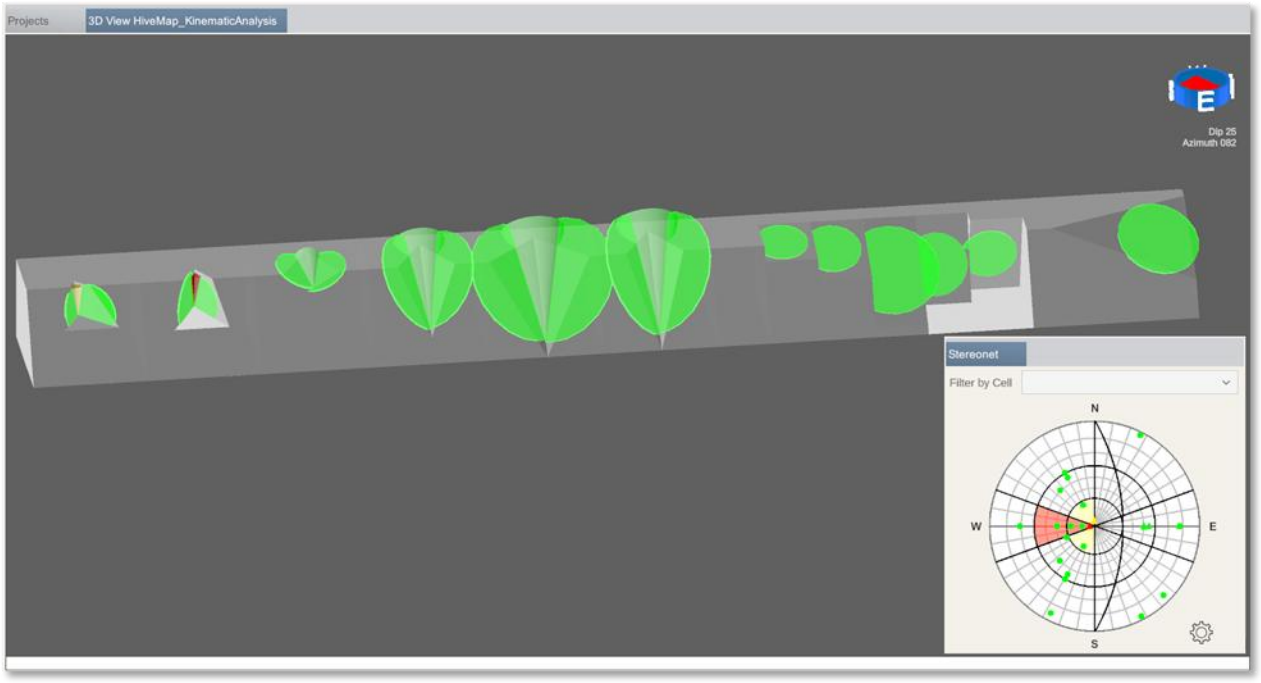
Flexural toppling: If a joint is going to cause a flexural toppling, it should be outside of slip limit (slope dip-friction angle) plane and it should be in the lateral limit. This area on stereonet displayed with red colour. Discs which are inside this area are coloured red and rest of the discs are coloured green.

Direct toppling: Once the disc intersections are calculated, this analysis can be run. If a joint intersection is going to cause a direct toppling, it should be inside the slope dip cone, it should be inside the lateral limits and dip into the slope. This area on stereonet displayed with red colour. Cones which are inside this area are coloured red.

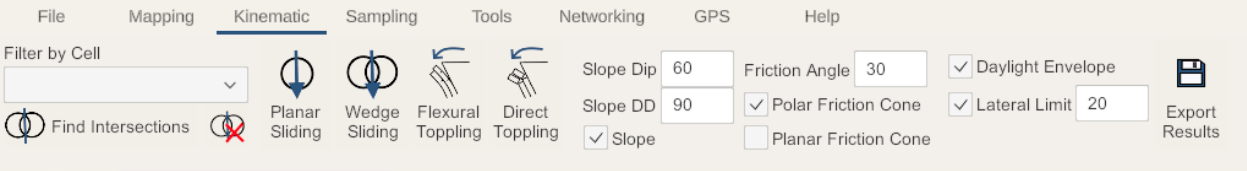
There is a secondary area where the wedge blocks topple at an oblique angle. This area is inside of the friction cone, outside of the lateral limits and dipping into the slope. This area on stereonet displayed with yellow colour. Cones which are inside this area are coloured yellow.

The rest of the cones are coloured green.

Table of Contents



After running analysis Export Results button is displayed. Click this button to export results into KinematicAnalysis folder. An image of the stereonet and a .csv file will be saved.

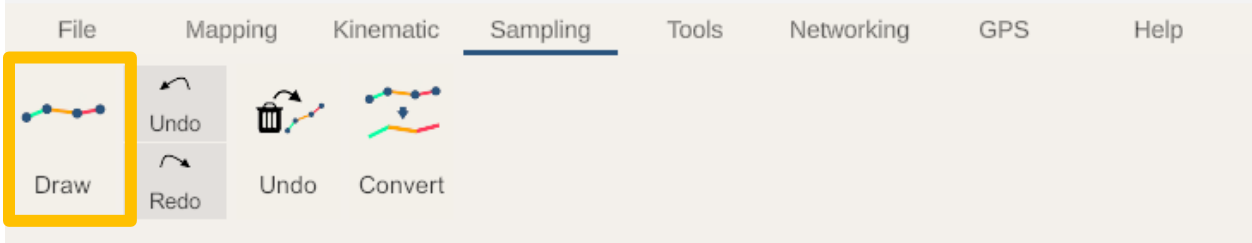


[Table of Contents](#)

3.6 Sampling

The Sampling tool may be used to support chip sample mapping.

Click Draw button.



Click the start and end points of chip sampling intervals on the mesh.

Use Undo and Redo buttons to undo and redo last node while drawing.

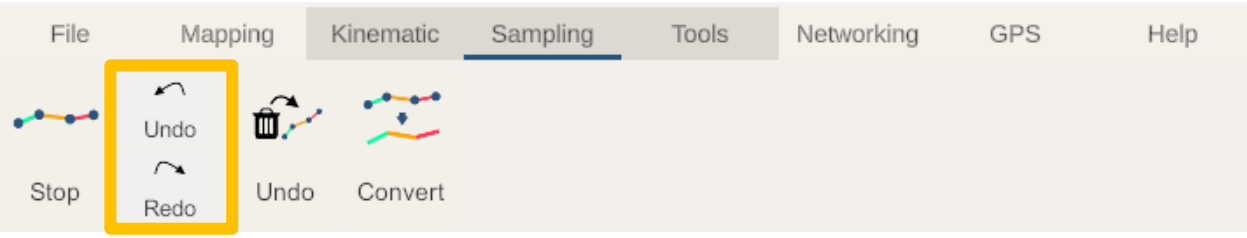
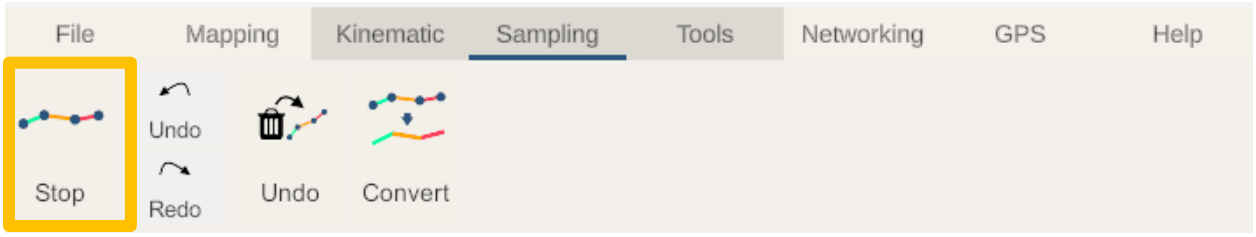


Table of Contents

Click Stop after finishing.



In the explorer panel under the Sampling branch, the sampling files are listed.

To edit the sampling data right click and select Edit Trace or left click the sampling polyline in the 3D View.

The Sampling Menu will be displayed, and the nodes of the trace polyline/polygon becomes visible.

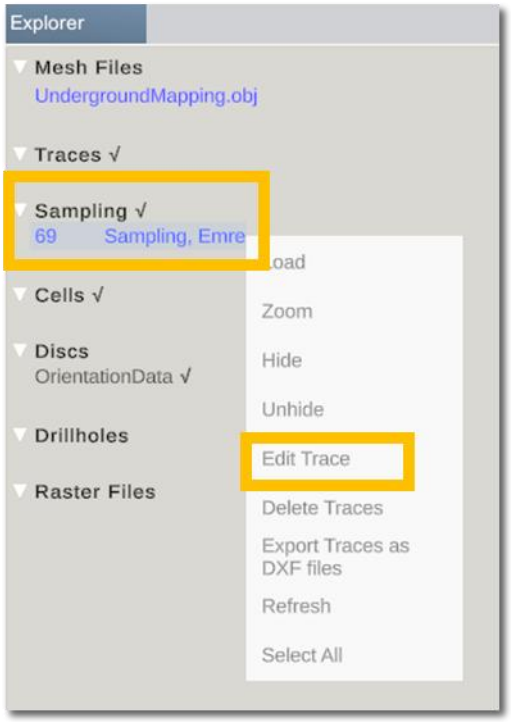
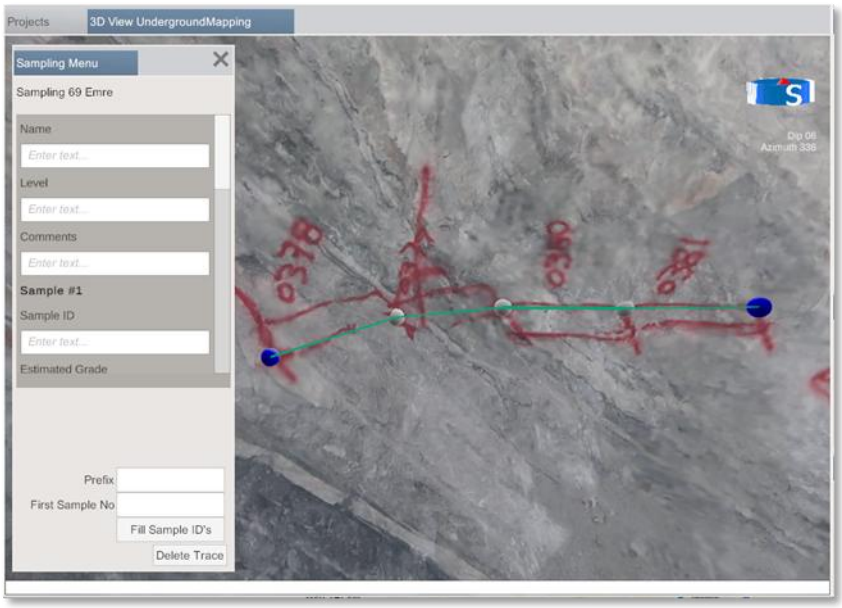


Table of Contents



Each segment of polyline represents a sample. Clicking a sample scrolls the form to that sample.

Sample ID fields on the form can be automatically filled. Type the Prefix and the sample number of the first sample. Click Fill Sample ID's.

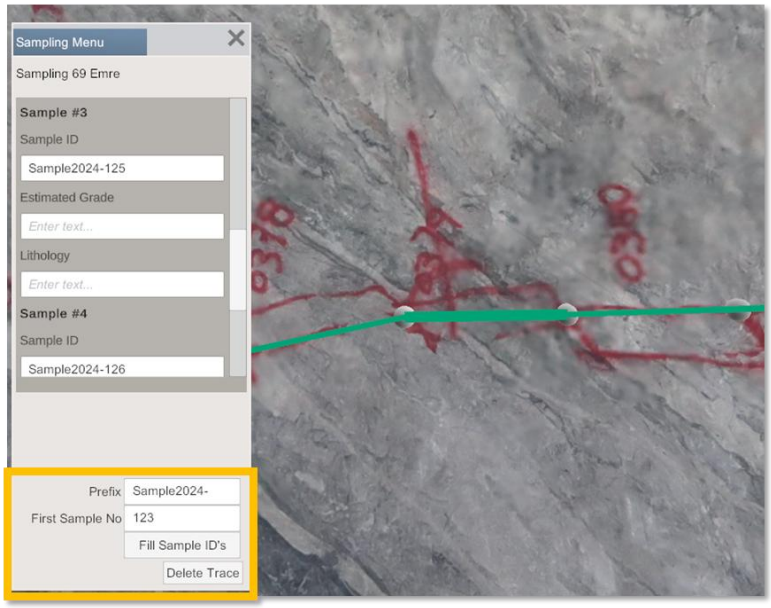
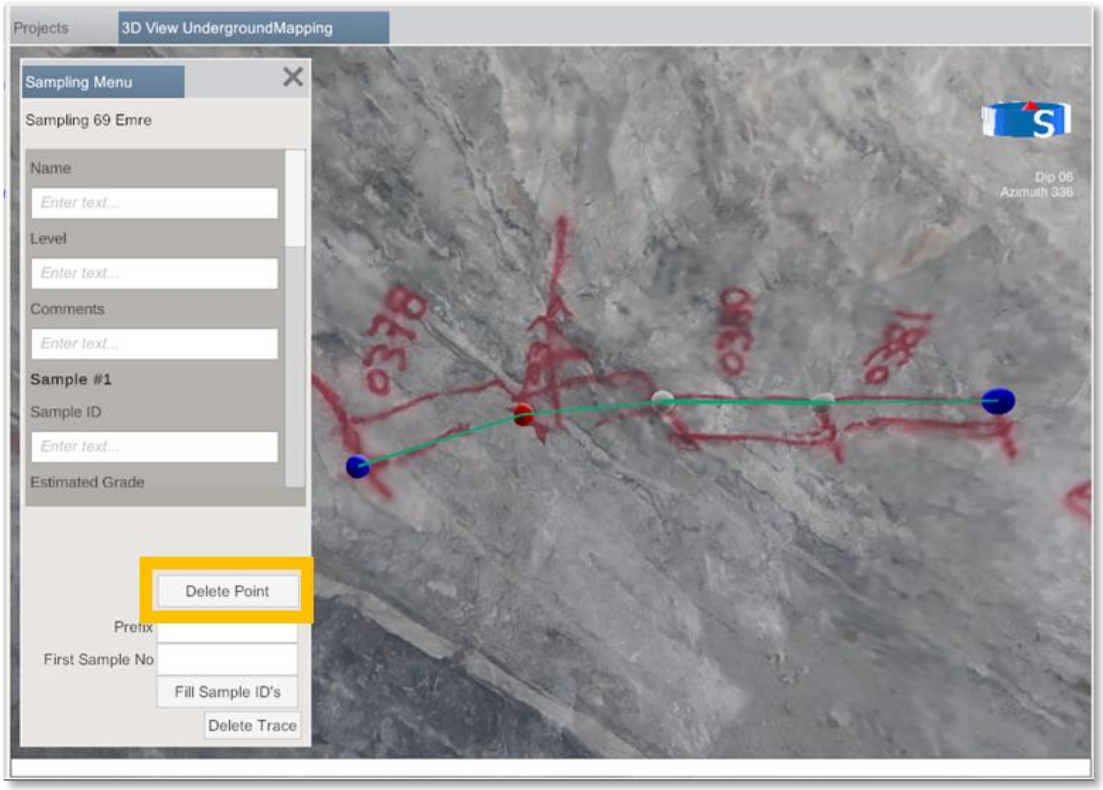


Table of Contents

Click one of the white nodes. Node will become red and, on the menu, Delete Point button will become visible. Click this button to delete the selected red node.



Click the mesh and the red node will move to the clicked position on the mesh.

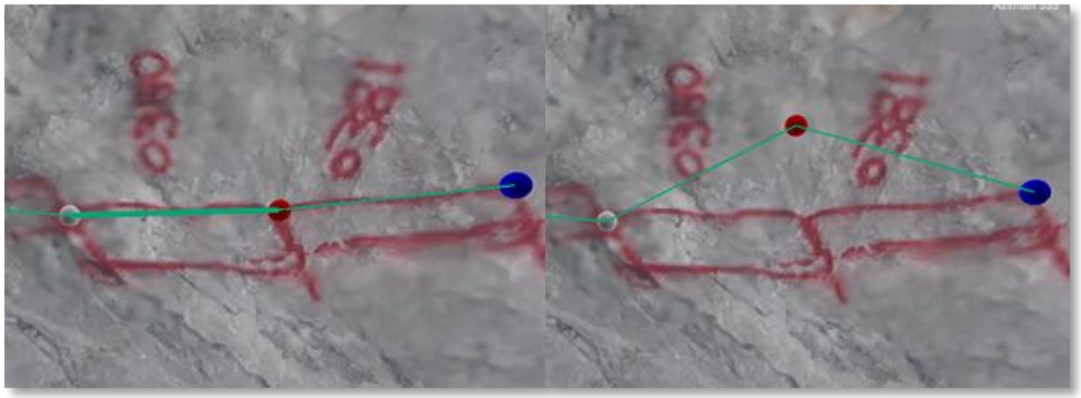
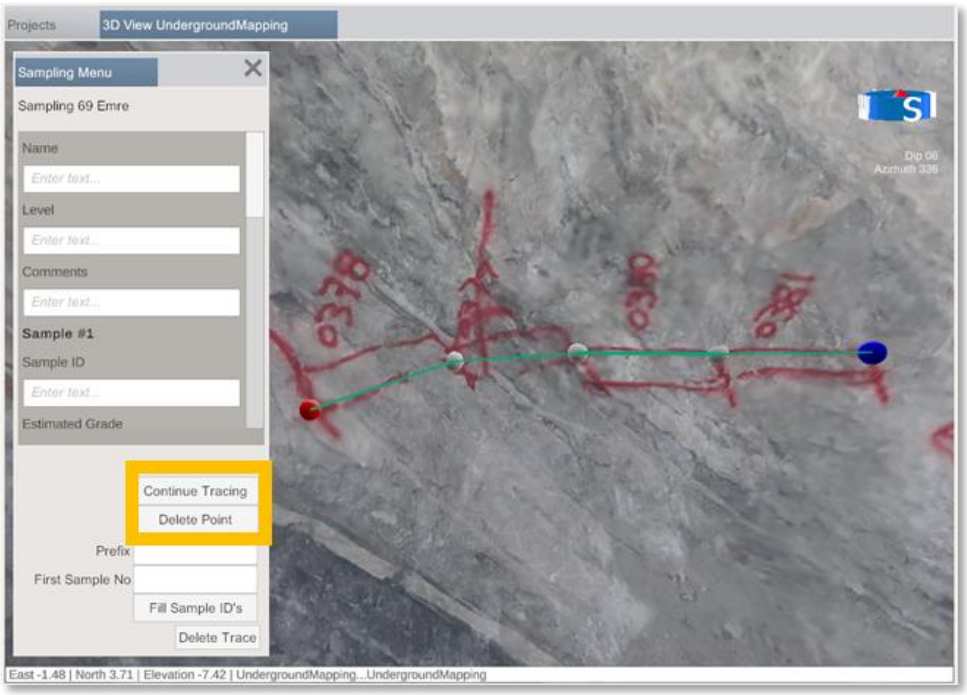
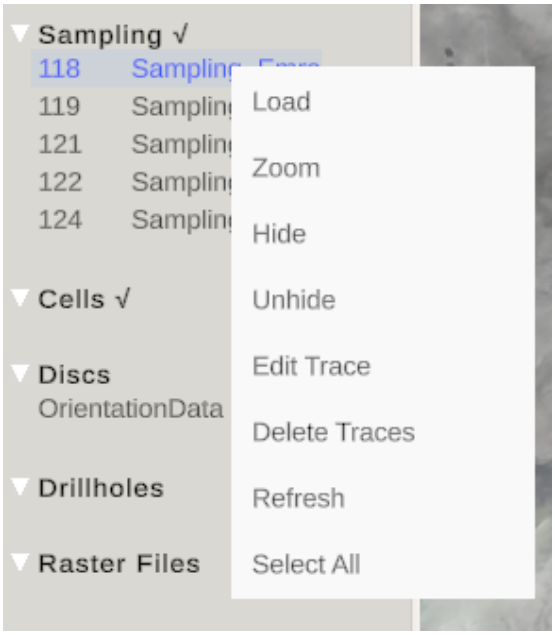


Table of Contents

Click the first or last node which is purple in colour. The node will become red, and, on the menu, Delete Point button and Continue Tracing button will become visible. Click continue tracing. This will close the Sampling Menu and hide the nodes and tracing will continue from the selected node.

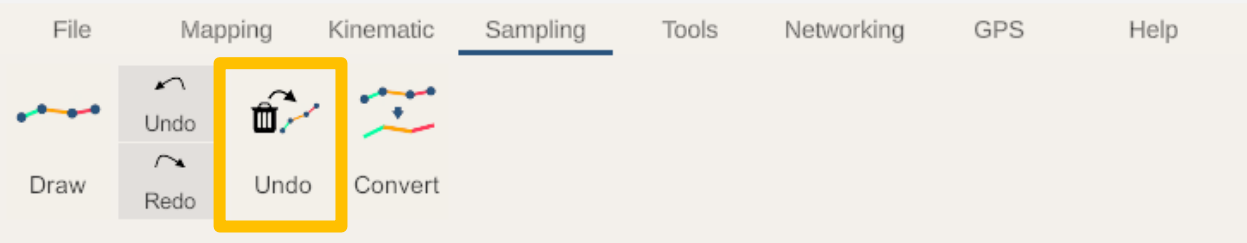


In the explorer panel, the right click menu includes options to zoom to a trace and hide/unhide traces. Besides editing a trace, multiple traces can be also deleted.



[Table of Contents]

Undo button undeletes the latest sampling. Deleted sampling files are stored in the Recycle Bin folder within the project folder.



Convert button converts sampling trace to a drillhole and saves into SamplingDH folder as .csv files.

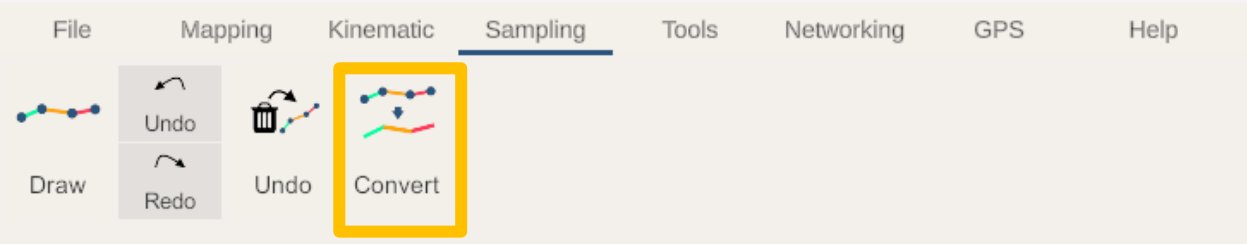
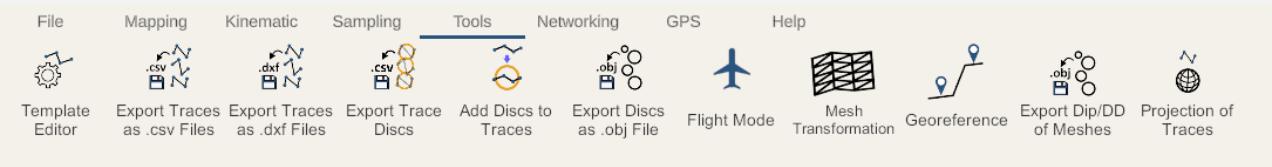


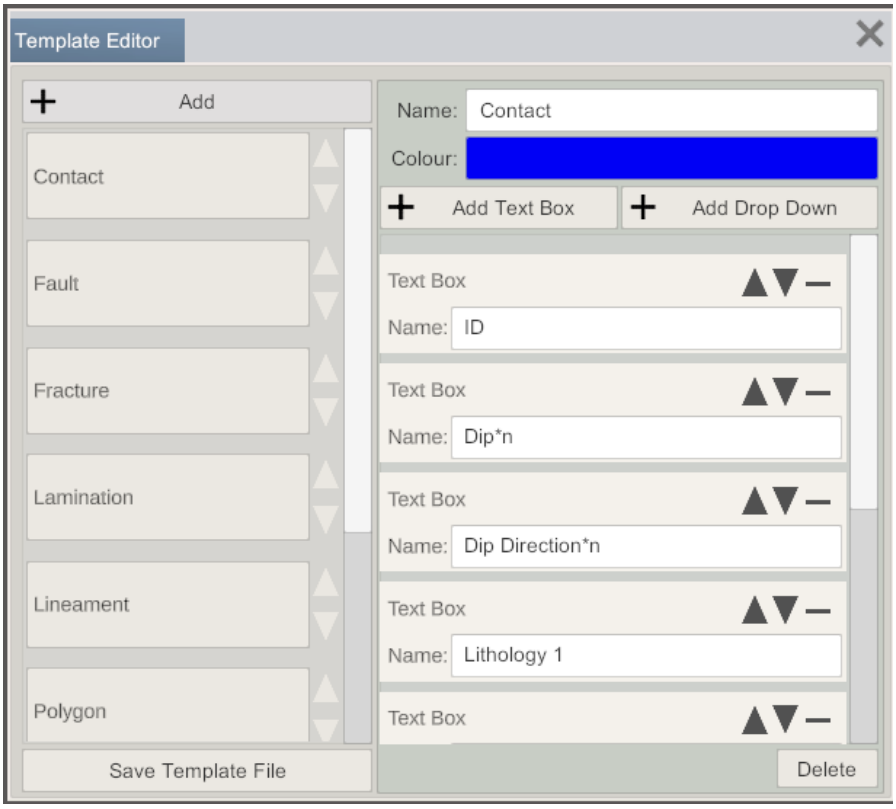
Table of Contents

3.7 Tools

The Tools menu includes various functionalities.

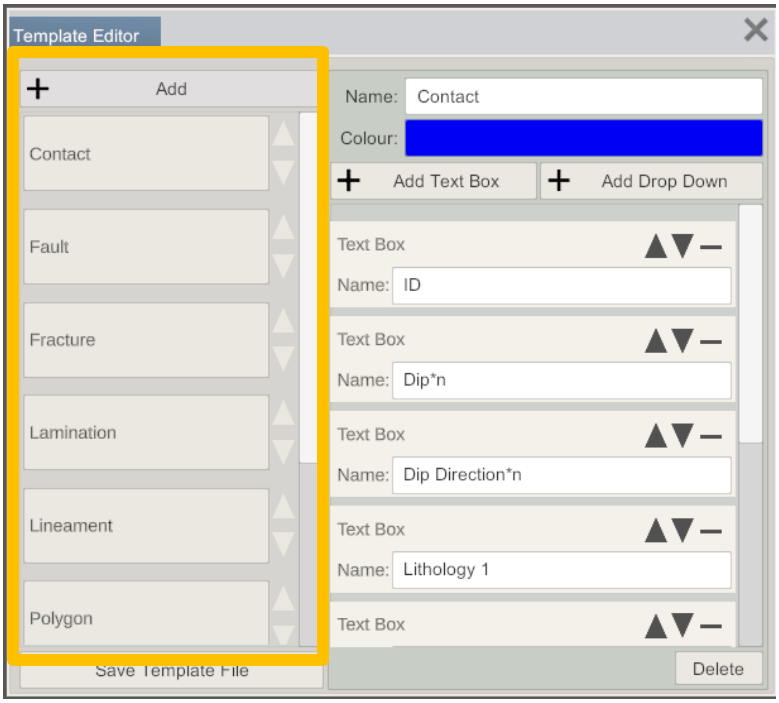


Template Editor: Trace structures and their attributes, cell mapping attributes, and sample attributes are customized in Template Editor.

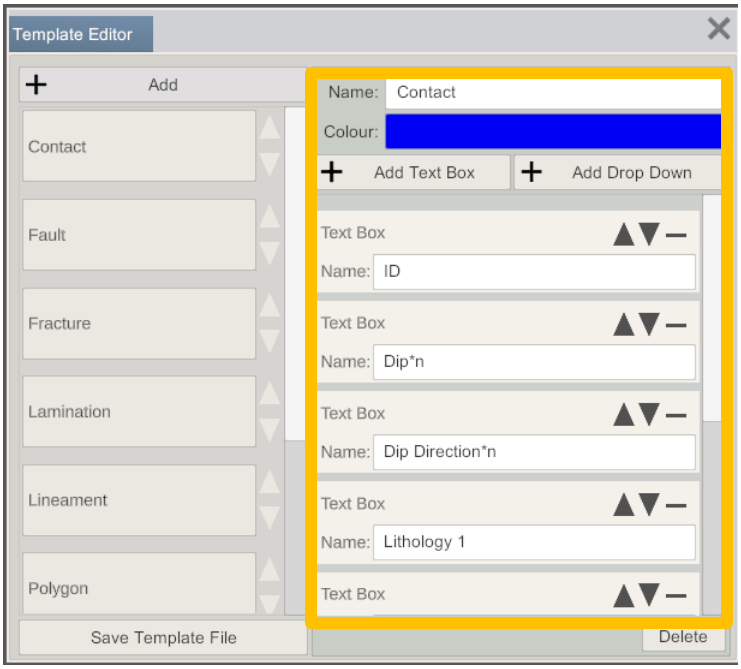


[Table of Contents](#)

Structures are listed in the left panel. Click up and down buttons, which are placed next to each structure button, to change the order of the structures. Click Add button to add new structure.

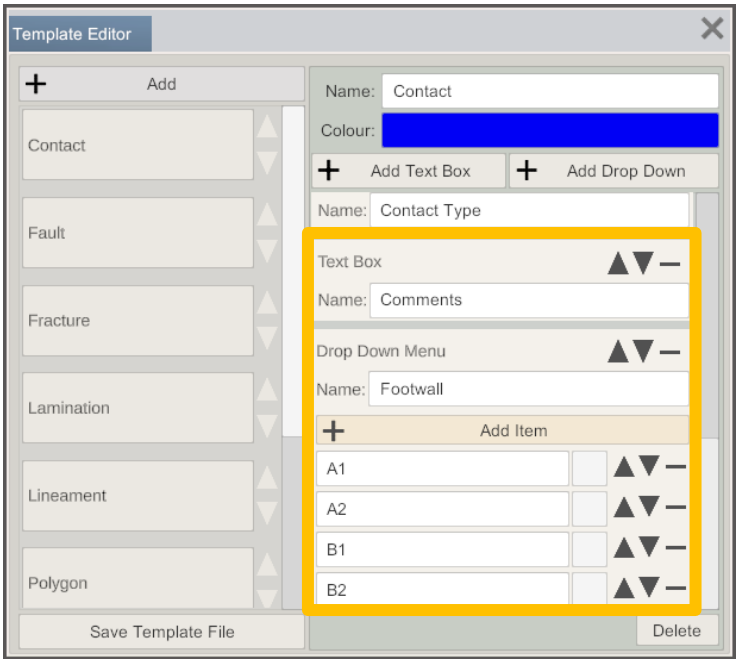


Click a structure to display the structure form on the right-side panel. On top of the panel, name and colour of the structure item can be changed. Click Add Text Box button to add a new text box to form and click Add Drop Down to add a new drop down list to the form. Delete button, which is located on the bottom of the panel, removes the structure.



[Table of Contents]

Click the up and down buttons to change order of the items on the form. Click – button to delete an item. Click Add Item button to add a drop-down item to drop down list. Click up and down buttons to change order of the list items and click the – button to delete them. Click the square box to change the colour of list item.



Export Traces as .csv Files: For each structure type a .csv file is generated. Polyline/polygon and attributes of the related traces are saved in these .csv files, which are located in ExportedTraceData folder of the project.

Export Traces as .dxf Files: Polylines/polygons of each trace file are saved as a seperate .dxf file in the ExportedTraceData folder of the project.

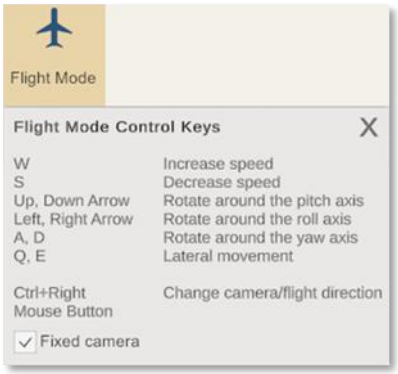
Add Discs to Traces: This will add a disc to traces which does not have a disc.

Export Trace Discs: All trace discs are saved into TraceOrientationData.csv in ExportedTraceData folder in the project folder.

Export Discs as .obj File: Discs are saved as a mesh file in .obj file format. This file can be found in the project folder.

Flight Mode: When the flight mode is activated, the camera in the 3D View can be moved with keyboard. Control keys are listed in the drop-down menu. This may be useful for capturing videos for use in presentations.

Table of Contents



Mesh Transformation: Mesh Transformation menu lists the .obj mesh files of the project.

- Flip y-z, x-y axis: Some software export meshes with different axis for vertical or horizontal axis. These files can be fixed with this tool.
- Flip triangles: This changes which side of the mesh is front, and which is side is back.
- Translate Mesh: Moves the meshes by adding values in the x,y,z boxes.
- Import new mesh files: Imports the processed mesh files.
- Delete proceeds mesh files: Processed mesh files are moved to the Recycle Bin folders.

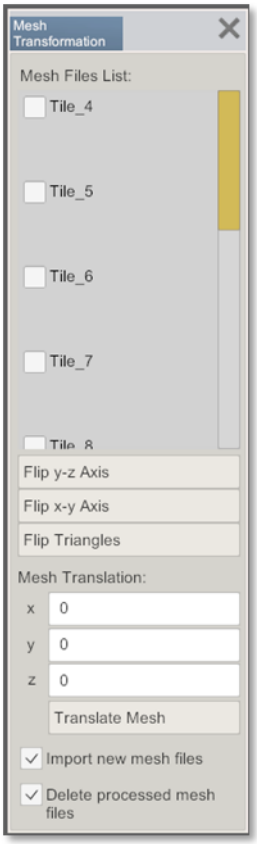
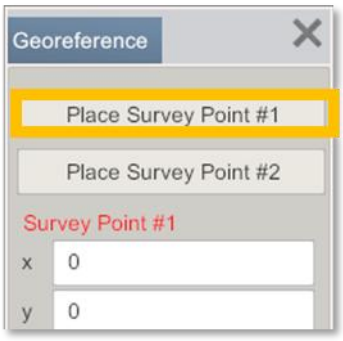


Table of Contents

Georeference: This tool georeferences mesh files, such as those from iPhone/iPad scans, which use local coordinates and have an incorrect northing direction. However, their scale and vertical orientation are accurate.

For georeferencing 2 survey points, or 1 survey point and the north direction is specified:

Click the Place Survey Point #1 button.



Click the survey point on the mesh, this will create a red sphere on that point.

Click Survey Point [Stop] button.

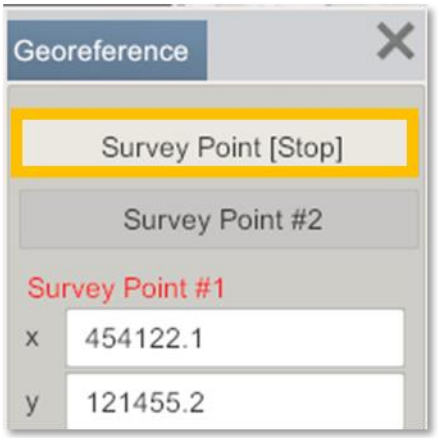
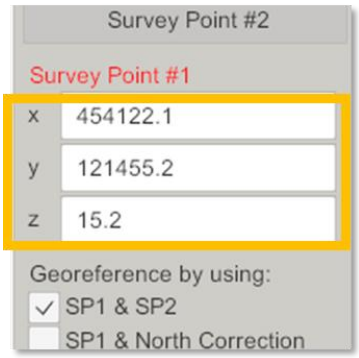
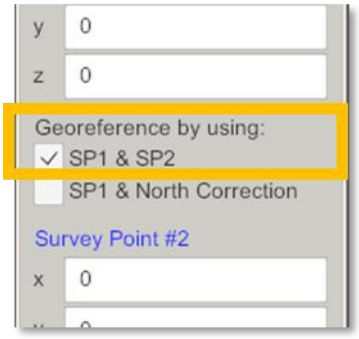


Table of Contents

Enter the Survey Point #1 coordinates.



If you have 2 survey points, select the SP1&SP2 checkbox.



Repeat the same process for the survey point 2.

If you have 1 survey point and you know the north direction, select the SP1&North correction checkbox, and enter the north correction which is the difference between the azimuth of the mesh and north.

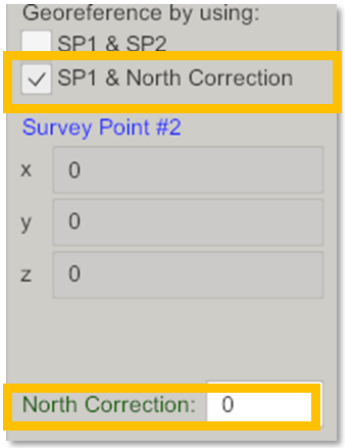
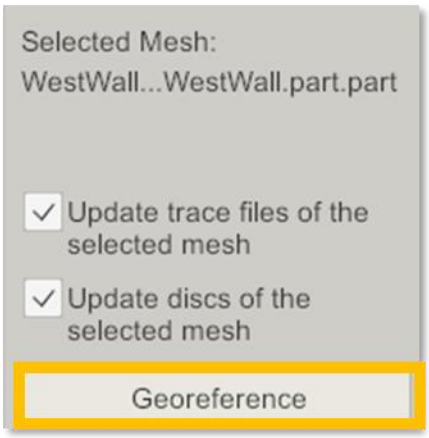


Table of Contents

Click the Georeference button.

If the mesh is mapped before it is georeferenced, the traces and discs will be not georeferenced either. If the Update trace files of the mesh and Update discs of selected mesh are selected, these files will also be georeferenced along the mesh.



Export Dip/DD of Meshes: It collects the following data from the meshes:

- Triangle centers: Orientation of the mesh triangles and position of the triangle center.
- Vertices: Orientation of the mesh vertices and position of the vertices.
- Inside triangles: Orientation of the mesh triangles and the 3 positions between triangle center and triangle corners.
- Average of each mesh: It finds the best fitting plane for the vertices of the mesh. This planes orientation and center.

Results are exported to the Exported_DD_Dip.csv file in the project folder.

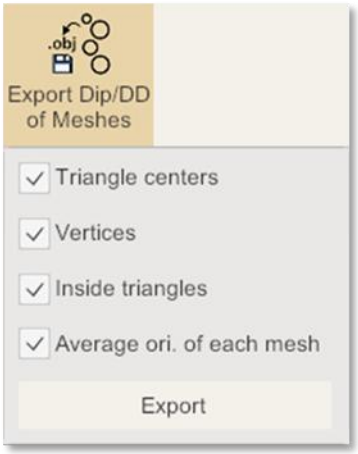


Table of Contents

Projection of Traces: This tool changes the coordinates system of the georeferenced traces to another coordinate system.

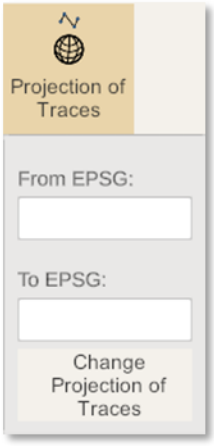


Table of Contents

3.8 Networking and Collaborative Mapping

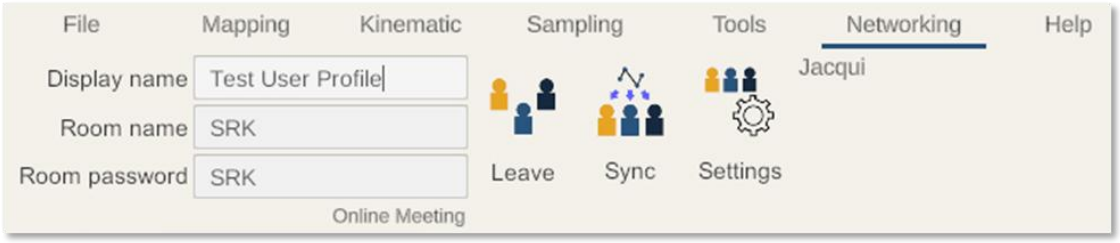
In HiveMap users can collaborate remotely. They can do virtual site visits where they can map virtual outcrops together. They can also do audits where they can review and edit the mapping.

During online meetings, screens are not shared. Mouse location and real-time mapping data are shared.

Each user should have the same mesh files, drill hole data and raster files before the meeting. All the participants should enter the same Room Name and Room Password to meet in the same room.

Display Name should be unique to each user.

Click the Join button to start/join the online meeting. Participants are listed in the right side of the menu, under Networking.



During the meeting in 3D View, each participant's mouse cursor is displayed with their display name.

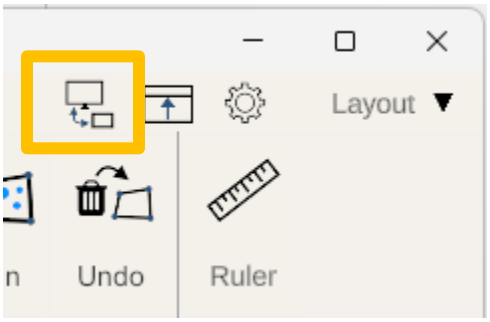
Table of Contents

All tracing data that is created during the meeting is shared between the participants in real-time. The Sync button syncs the traces which are created before the meeting.

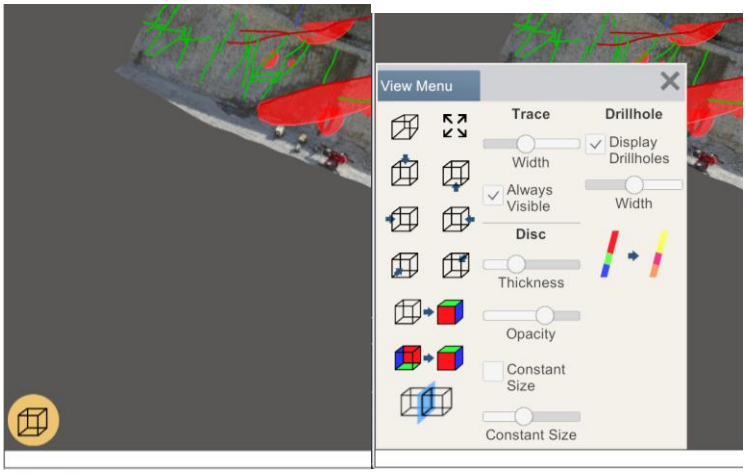
Click the Leave button or close the HiveMap to leave the meeting.

3.9 Tablet Mode

By clicking the Switch button on the top left corner, you can switch between tablet mode and desktop mode. Tablet mode changes scale of the user interface to make menu items larger so the users can easily use their fingers instead of a mouse to interact with them. The scale for tablet mode can be adjusted in the Settings Menu. The layout is switched to the Tablet mode.



The View menu is hidden and can be displayed by clicking the View Menu button on the lower left corner.

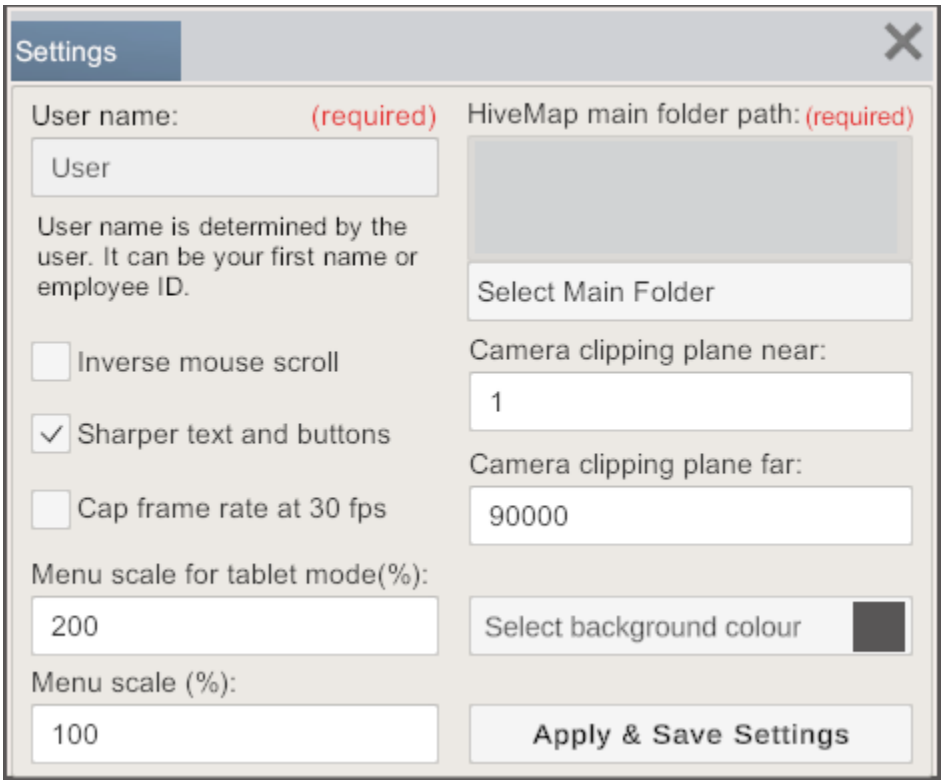


Single finger behaves like left mouse button. In the Explorer panel holding your finger longer on the screen will open the right click menu. In the 3D View panel, 2 fingers are used for zooming in/out and panning. 3 fingers are used for rotation.

Table of Contents

3.10 Settings

To open the settings menu, go to the Help menu and click the Settings button or click the settings button on the top right corner of HiveMap windows.



Username: The username is included in the trace file names and in the author section of the files. If a trace file is edited, the username of the editor is saved in the file. In orientation data files, the authors username is also saved for each disc.

Main folder: All HiveMap project folders will be created in this folder. To update the main folder location, click on Select Main Folder. The Select Folder browse window will pop up. Select a folder location where all HiveMap files can be saved.

[Table of Contents]

Inverse mouse scroll: Mouse scroll direction for zooming in 3D View changes.

Sharper text and buttons: On default it should be checked. If there is performance issue with your computer, this can be unchecked to improve it.

Cap frame rate at 30fps: On default it should be unchecked. If there is performance issue with your computer, this can be checked to improve it.

Menu scale (%): Default value is 100%. Increasing this value will increase the user interface elements and decreasing it will make them smaller.

Menu scale for tablet mode(%): User interface scale during the tablet mode.

Camera clipping plane near: Default value is 1. Camera cannot see anything closer than this distance. Decreasing this value below 1 is not suggested as it might decrease the accuracy of the mouse.

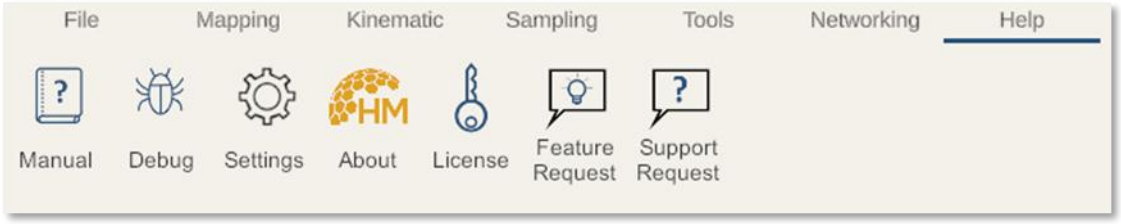
Camera clipping plane far: Camera cannot see anything further than this distance.

Select background colour: 3D View background is adjusted here.

Table of Contents

3.11 Help

Within the Help section, users can access the user manual, the settings, about details, license ID, the feature request, and the IT support ticket forms.



The Manual button opens the link of the .pdf manual file.

The Debug can be accessed by users when troubleshooting issues with Help Desk support.

The Settings button opens the settings menu.

The About button opens the HiveMap about menu. HiveMap version number is displayed here.

License button opens License menu (link).

The Feature Request and Support Request buttons open the forms to contact HiveMap support.

4 Support

4.1 Reporting Issues

If users encounter any technical issues with HiveMap, please submit an IT Support Request Form. Help Desk support will follow up with clients for each ticket submission.

4.2 Requesting Product Feature Enhancements

If users would like to request a product feature enhancement, please complete the Product Feature Request Form. Help Desk support will log each product feature request submission, and these will be reviewed periodically as product development progresses.

4.3 Other Inquiries

For any sales and or general inquiries (not tech / help desk related), please email Inquiries@hivemap3d.com.

4.4 System Requirements

To run HiveMap software, the minimum system requirements for Desktop are as follows:

- Operating System: Windows 10 / Windows 11
- Memory: 8 GB (Minimum), 16 GB (Recommended)

4.5 Resources

Users can access HiveMap related resources on the website Resources page: hivemap3d.com/resources

Resources available include:

- HiveMap User Manual
- HiveMap Training Data.zip files (contains 3 zipped sub-folders)
    - HiveMap_JointSetAnalysis
    - HiveMap_Training_Initial
    - HiveMap_Training_Interpreted
- Privacy policy
- End User Licensing Agreement (EULA)

4.6 Software Updates

When there is a new update for HiveMap, users can use Microsoft Store to update HiveMap. If “App Updates” are on in the settings menu of Microsoft Store, updates will be done automatically. If it is “Off” users can go to the “Downloads” tab and update HiveMap.