POINT CLOUD PROCESSING

Software:

Power GEOPAK V8i (SELECT Series 4)

TerraScan Version 017.001

Contents

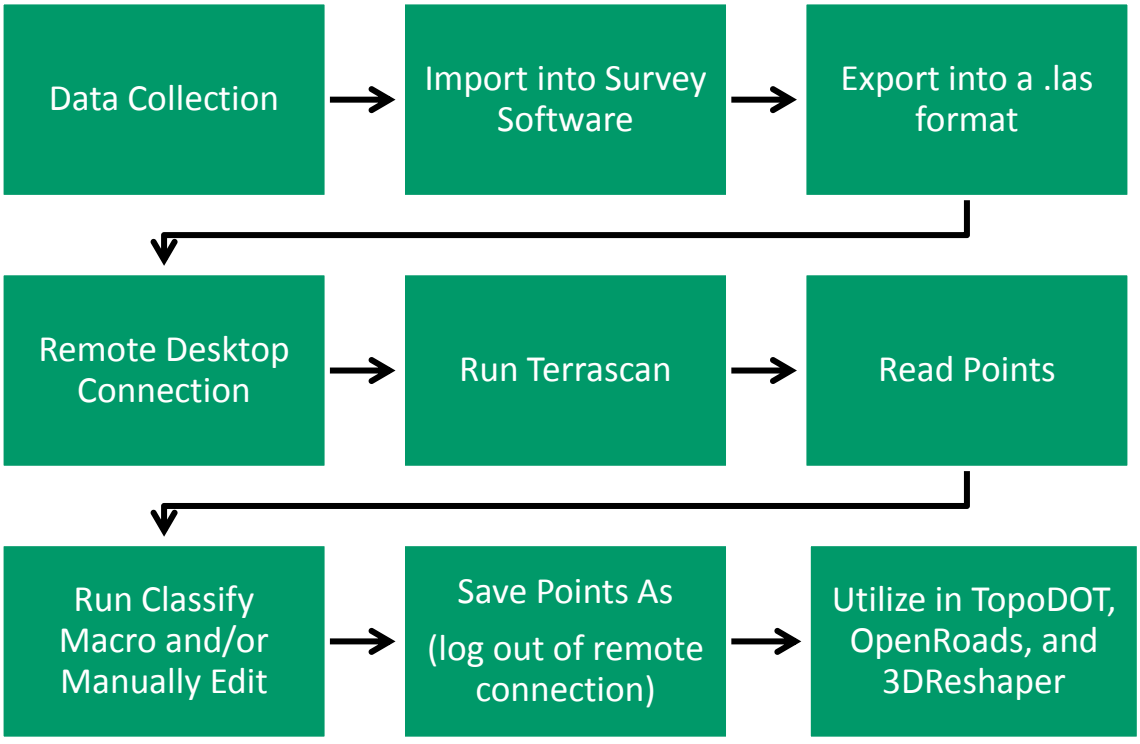
Intent

The intent of this document is to provide insight to the process and workflow behind editing and utilizing terrestrial LiDAR data in the form of a .las.

Preparation

This document assumes that the data is preprocessed and exported into an open source native point cloud format. For example, using the Leica ScanStation, the data is exported out of Infinity in a .las format (used in this demonstration). Additionally, this document assumes the user has access to remote desktop connection (to utilize a license of Terrascan).

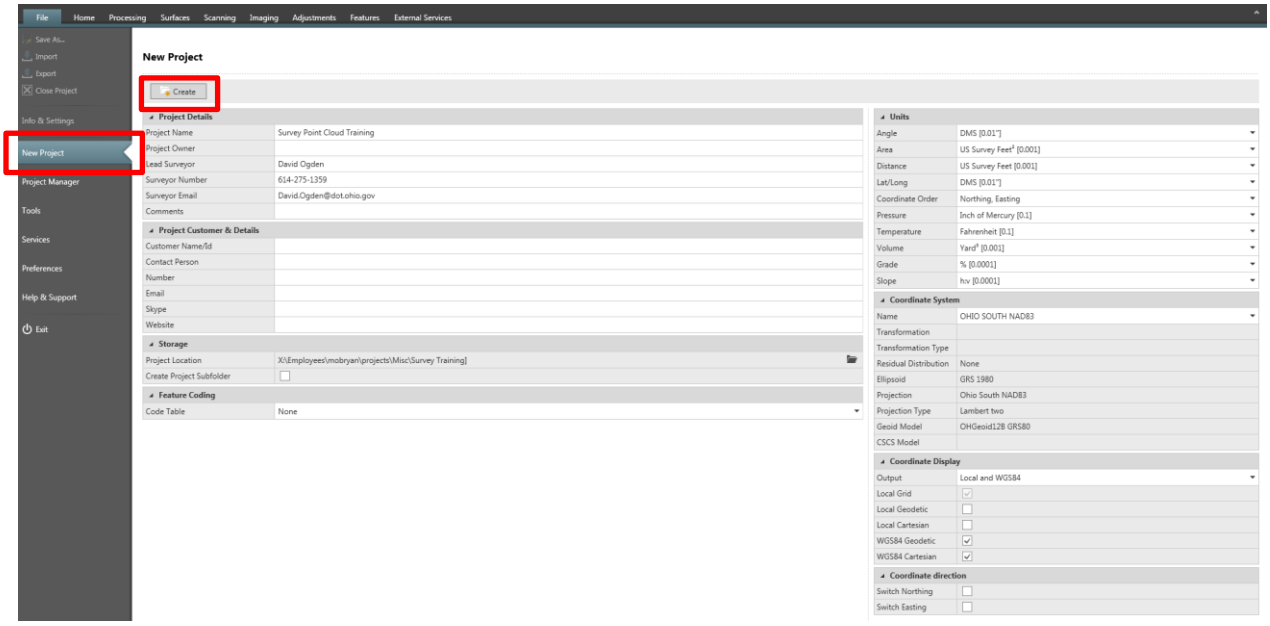
Workflow



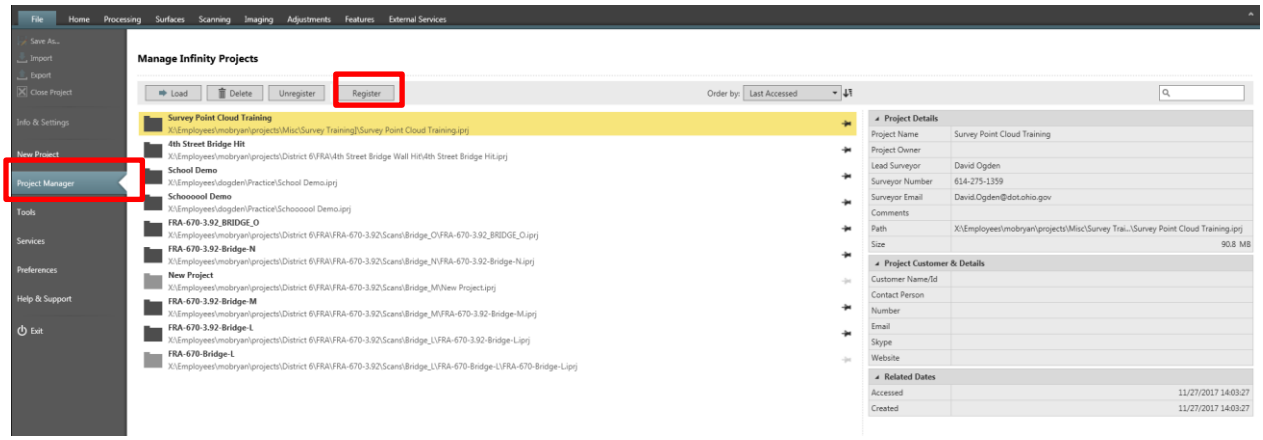
Leica Infinity

Opening a Project in Leica Infinity

Option 1: Open Infinity and select New Project. In this window, you will name your project and choose your folder destination. You will also want to define your units and coordinate system at this time. Once everything is filled out, click Create.

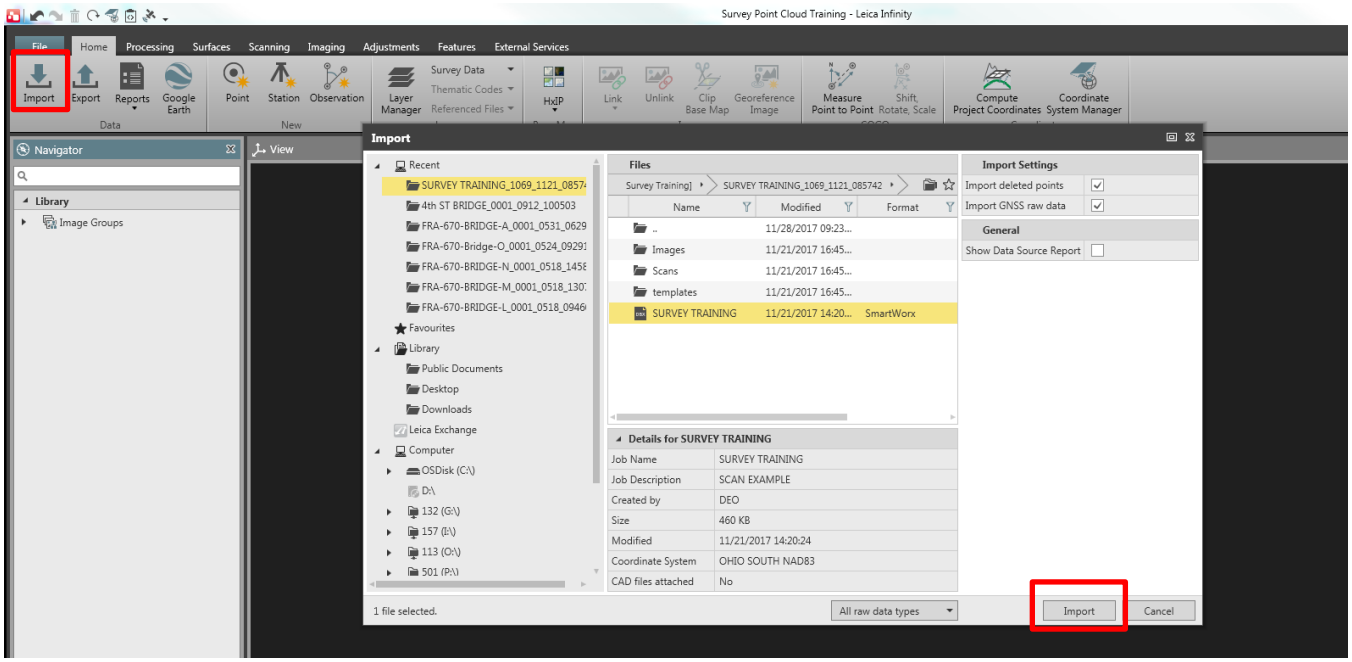


Option 2: If the project was previously created, you must register the project before you can open it. To do this select Project Manager, then select register and browse to your .IPRJ file. Your project will now be in the list of projects under Project Manager.



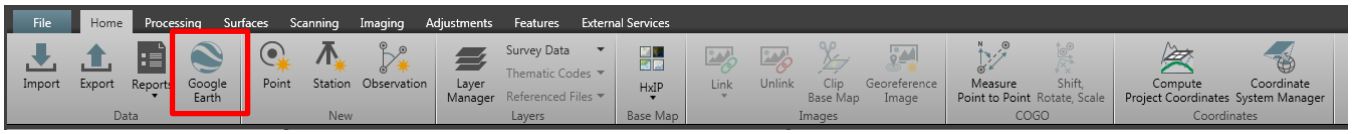
Importing Raw Data

Under the Home tab select the Import button, a new window will appear. Browse to wherever you placed your raw data. Select the DBX file (Format = SmartWorx) and click Import on the dialogue window. This will bring in all your control as well as all your completed scans.



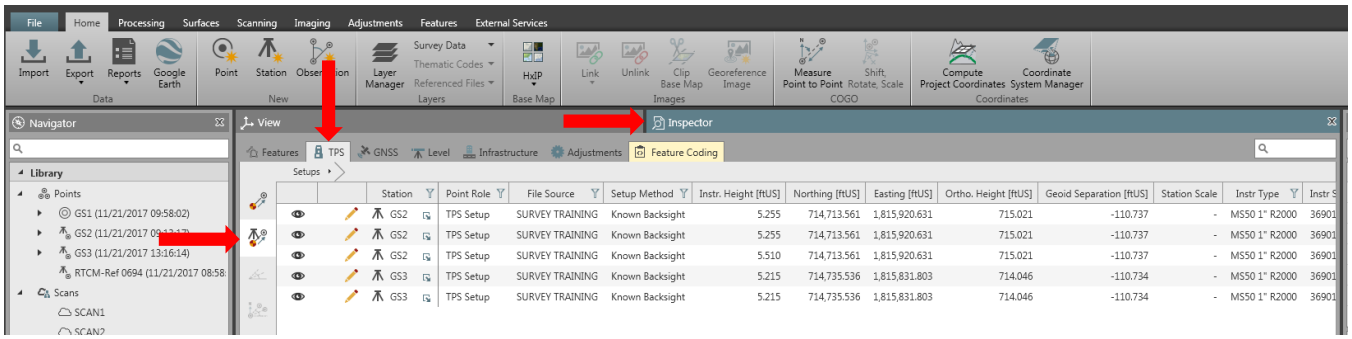
Google Earth Check

Much like Trimble Business Center, Infinity allows you to quickly view your data in Google Earth. This is typically the first step once you import your raw data to quickly verify that your data is in the correct location. Under the Home tab click Google Earth

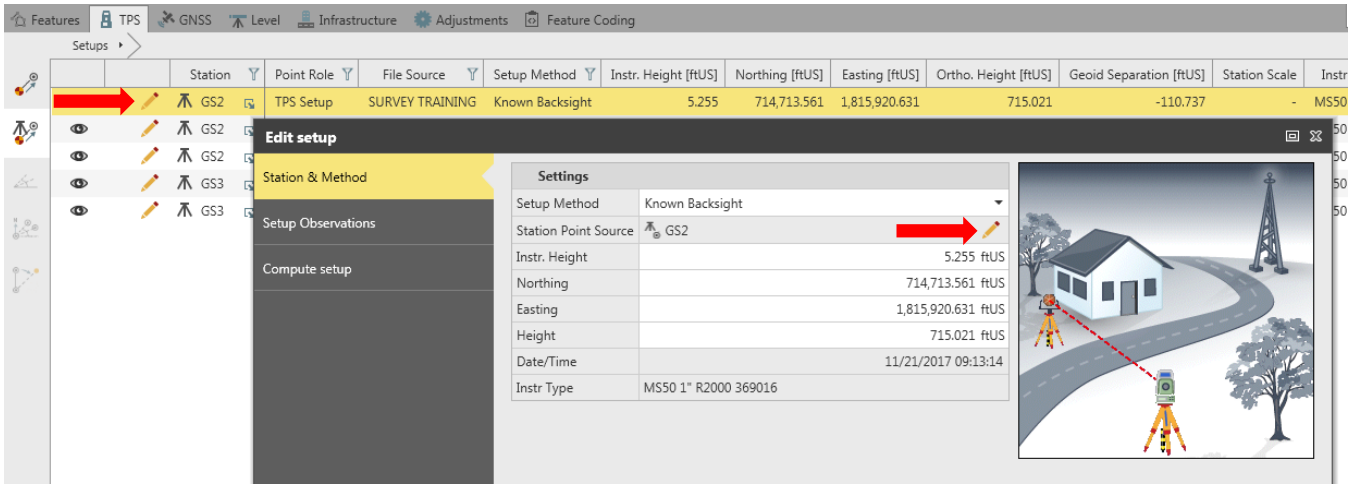


Fixing a Wrong Setup

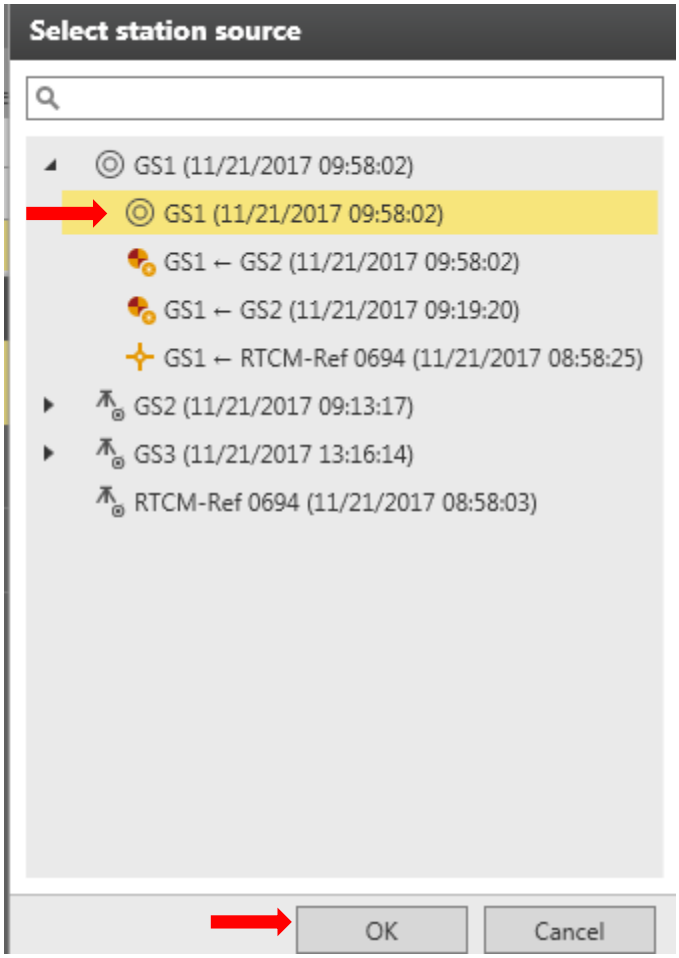
From this data, you can see that one of the scans if rotated 180 degrees. This is due to choosing the wrong setup point and wrong backsight point. For this example, the first scan was incorrect. We selected our setup point as GS2 and backsight point as GS1 however, they should have been reversed. To fix this, click the inspector tab and then the TPS tab, make sure the TPS Observations by Station Source is selected. This will list all your setups including your "backsight checks".



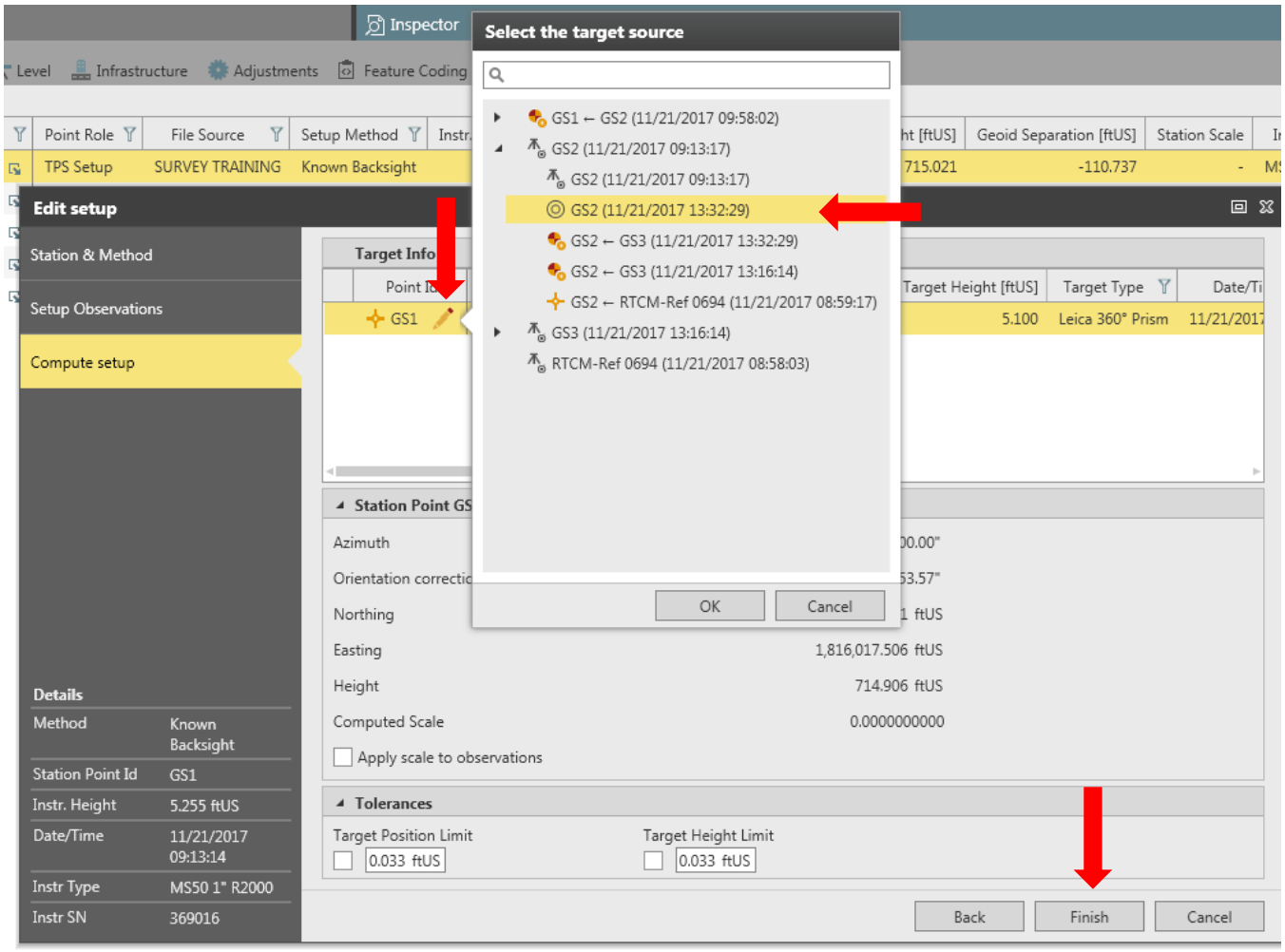
Next to the first wrong setup select the edit (pencil icon), a new window opens. Our current Station Point Source is GS2 when it should be the control point GS1. To fix this, select the edit (pencil icon)



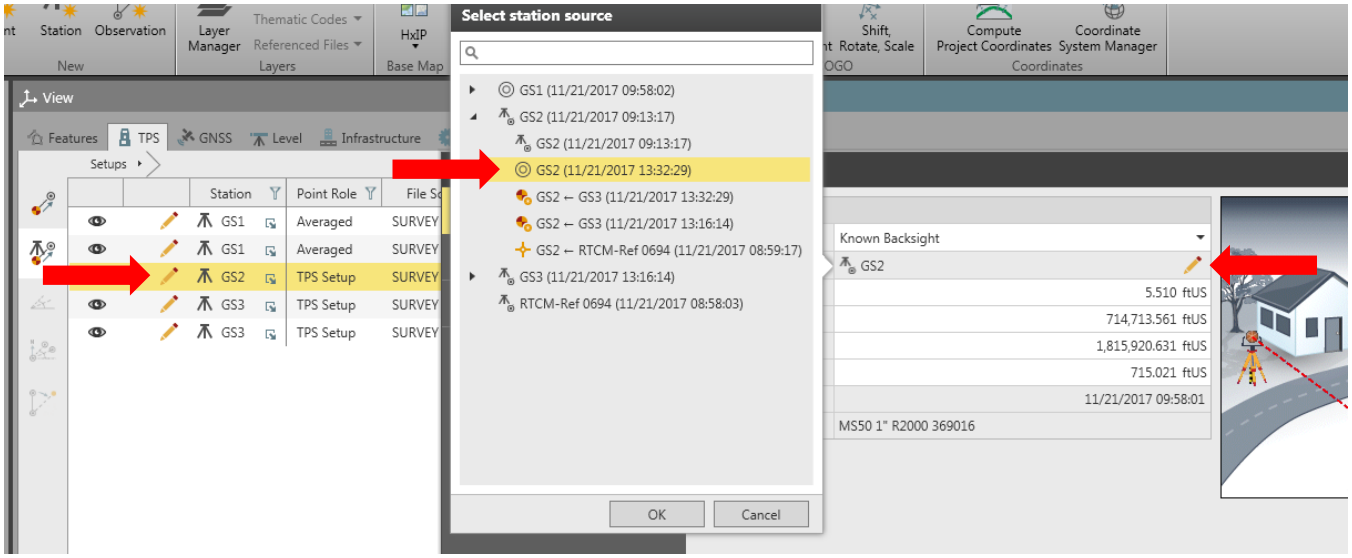
A new window appears, select the appropriate control point (GS1).



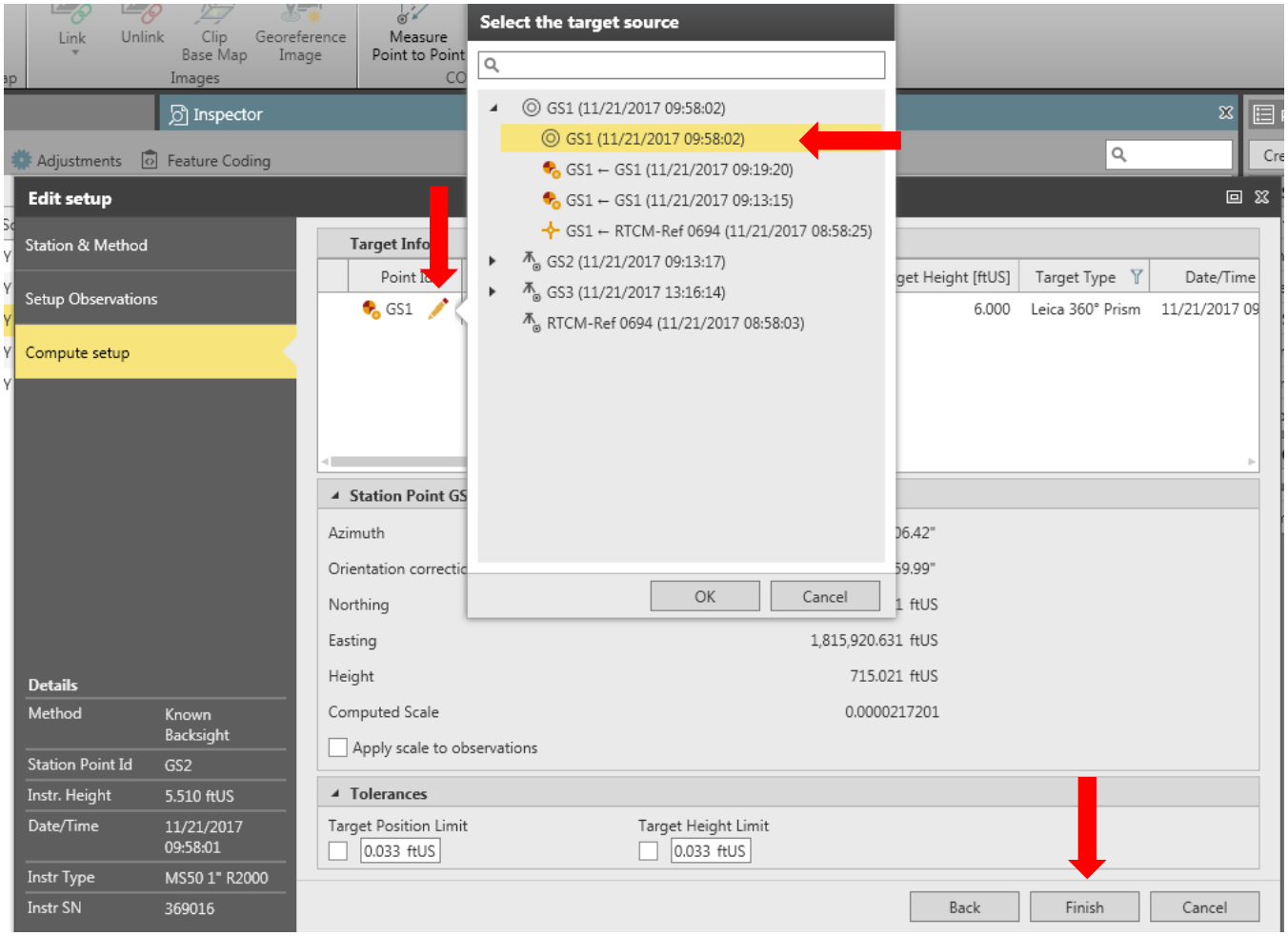
Click next in the Edit Setup window. This is the Setup Observations window. This is where you can select a certain point to use for the setup. For this data set there is only once choice so click next at the bottom of the window. You are now in the Compute Setup window. In this window, select your target point. For this setup, we should have chosen GS2 as our backsight. To fix this select the edit (pencil icon) and choose the correct control point GS2. Click Finish. The next Setup Station is from when we checked our backsight. We must also fix this setup using the same method above. Once completed this will adjust your scan and rotate it correctly.



Since we had multiple Total Station setups label GS2 we must fix the second scan as well. We will be replacing the total station setup with the correct control points. The steps are similar to the previous 2 setups however you are choosing GS2 as the setup point and GS1 as your known backsight point. Like before click the edit button next to GS2 (third one down) in the Inspector Window. Then click the edit button next to Station Point Source within the Edit Setup Window. Now select GS2 as your control point.

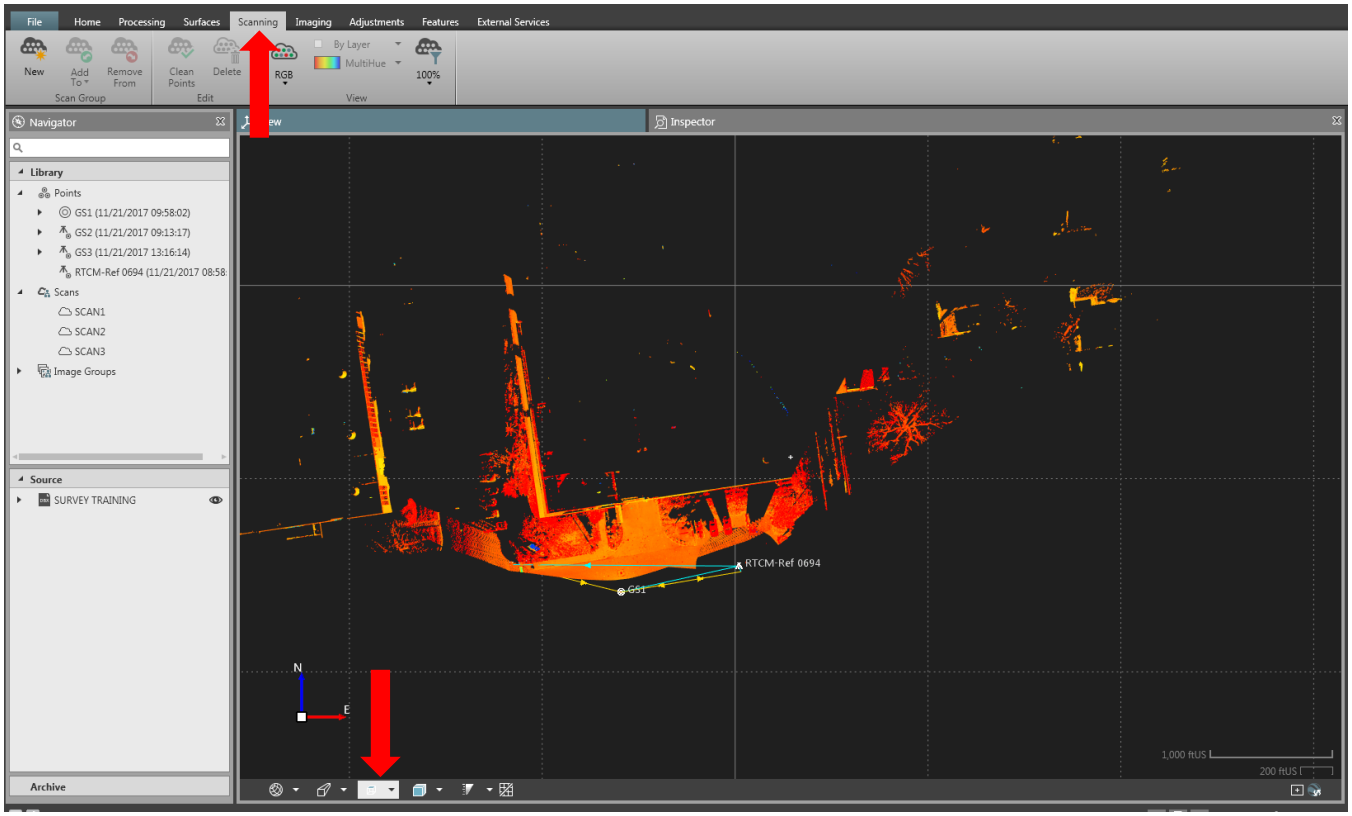


Click next in the Edit Setup window. This is the Setup Observations window. This is where you can select a certain point to use for the setup. For this data set there is only once choice so click next at the bottom of the window. You are now in the Compute Setup window. In this window, select your target point. For this setup, we should have chosen GS1 as our backsight. To fix this select the edit (pencil icon) and choose the correct control point GS1. Click Finish. Once completed this will adjust your second scan and rotate it correctly. Our blown backsight will now be fixed and we can proceed.

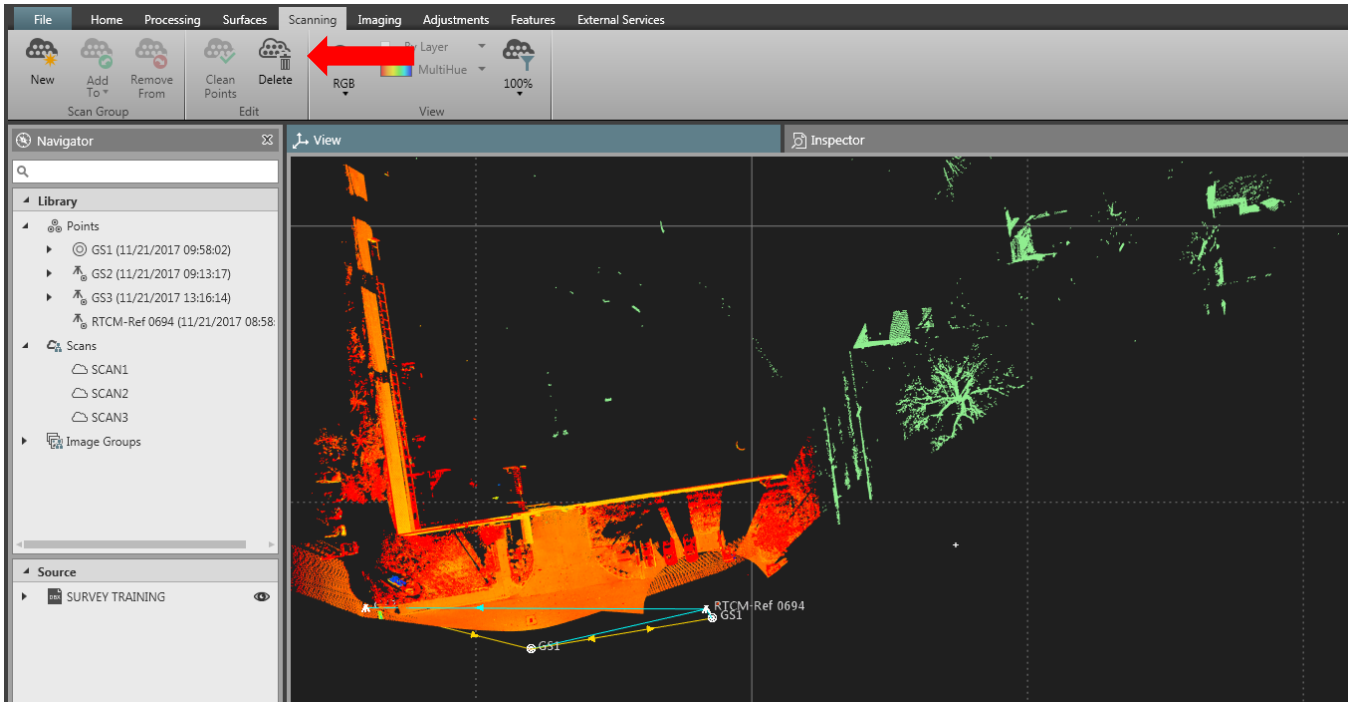


Preliminary Data Clean-Up (Use with Caution)

An easy way to reduce the size of your point cloud is to delete the unneeded points. For this dataset, we can see that there are point's way out from the building. To delete these points, click the scanning tab. As a note, you cannot zoom in and out when selecting your data because your selection window does not correspond. So, make sure you are zoomed in to what you want to see. You can also change the view that you are in by selecting the middle view icon at the bottom.

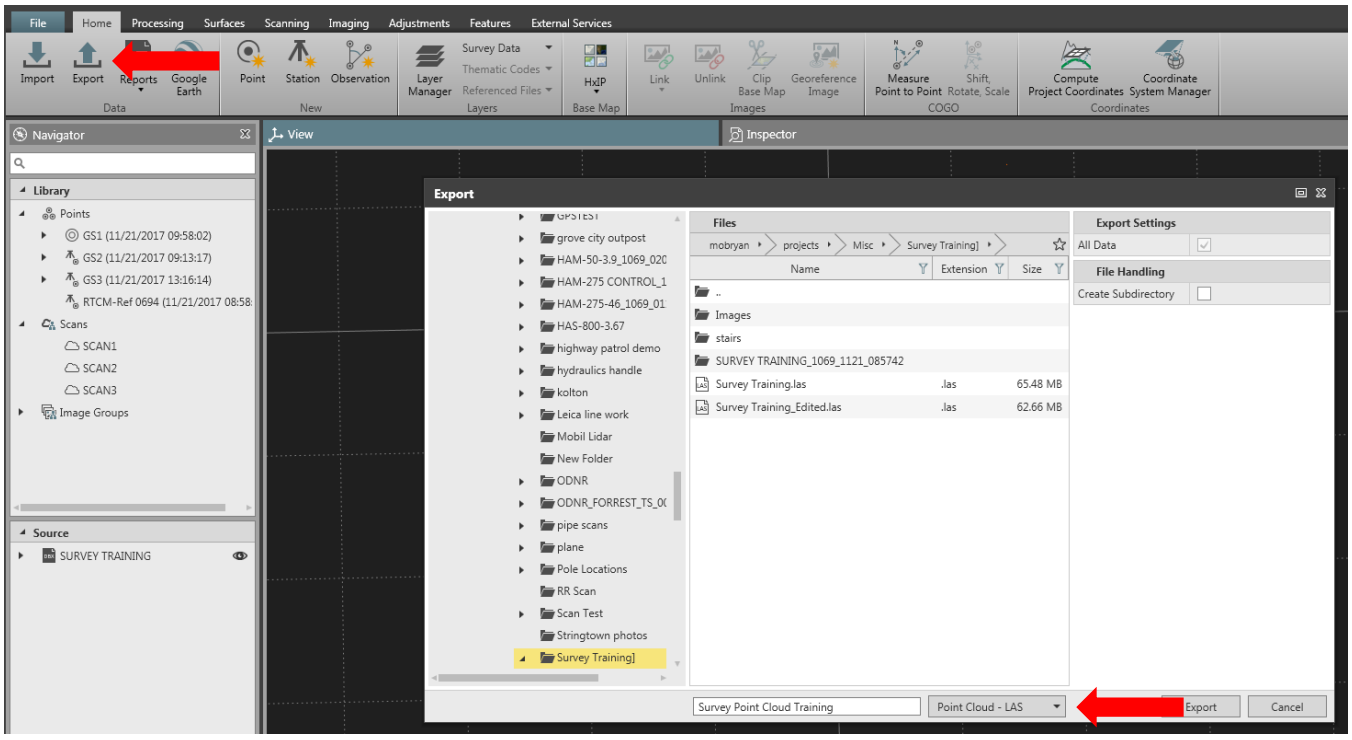


To select the data hold shift, click and drag around your section (It will turn green when selected). You can also hold shift and click to create a polygon around your selection. You can then click to cancel or click delete along the top ribbon to delete those points. This should only be used as a preliminary option. Other tools such as Terra Scan are much more powerful and accurate when dealing with editing point clouds.



Exporting to LAS

Once you have finished reviewing/editing your data you are ready to export a LAS. Under the home tab, select Export. A new window opens. Browse to where you want to store the Las file. Choose LAS as your format and click export. This will export all scans into one LAS file. You can also select part of the data using the selection process aforementioned. When you have, data selected you have the option to export all or export selection when you click the export button.



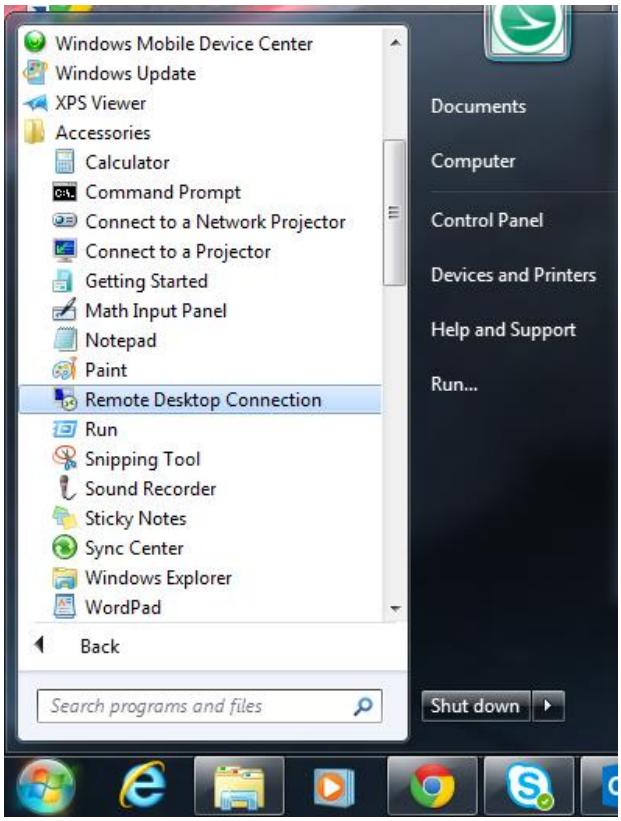
Remote Desktop Connection

To utilize the Terrasolid suite of products remote desktop connection needs to be used. The software is licensed on a single computer located in CADD and Mapping Services (due to how often the software will be utilized compared to the price). This poses a small challenge due to the fact only one person can be logged in at a time. Fortunately, the process takes 10 minutes or less so if by chance someone is remoted in the process should not take long. It is imperative that the user disconnects from the remote connection once the process is completed to allow other users the opportunity to use it

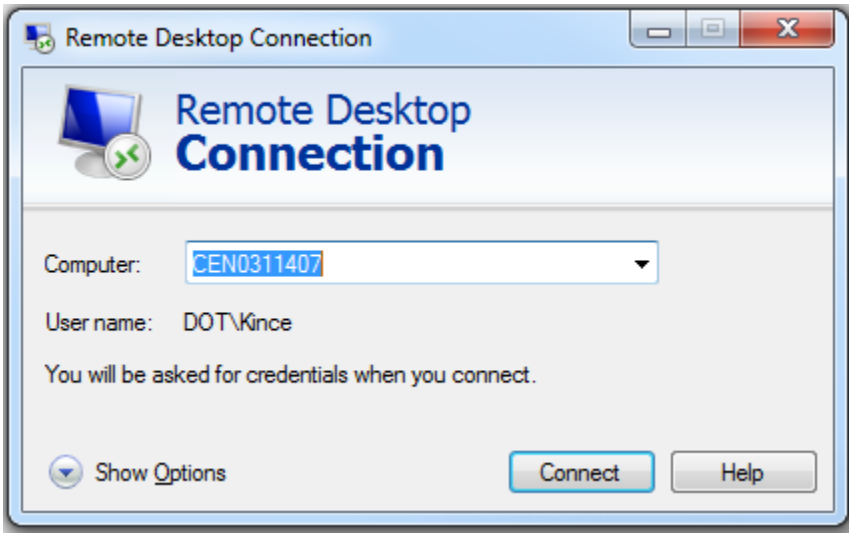
TerraScan Point Cloud Editing

Establishing the Remote Desktop Connection

Under the start menu click All Programs → Accessories → Remote Desktop Connection 



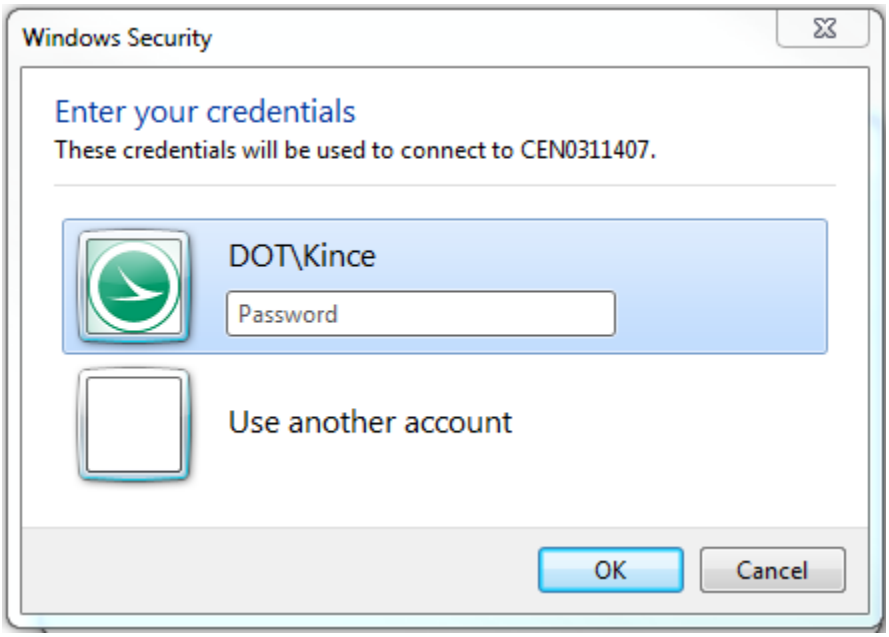
A dialog box will appear to locate the computer you would like to remotely connect to (see next page)



The computer name you are trying to remote into is CEN0311407 and your user name will appear in the section below.

Once selected click the Connect button.

A windows security dialog box will be displayed and the user will need to login using their ODOT credentials (same as the user and password to log into the computer)



Click OK to continue.

You are now logged into the computer and an ODOT Computer & Internet Use Policy screen will appear click ok to continue.

Note: If you are using dual monitors the computer only remotes into one screen the other screen will be the user's computer.

Opening Microstation

Open Microstation (either standard is fine a .dgn won't be created during this process).

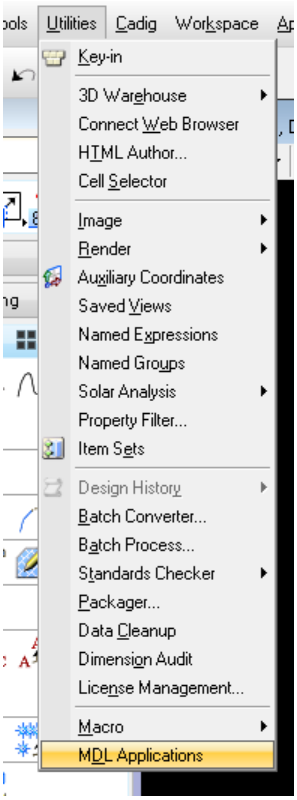
Open a generic 3D design file (the same file can be used repeatedly because no elements will be created in the .dgn)

When opening Microstation a popup window might open with a message about precision, this window can be ignored.

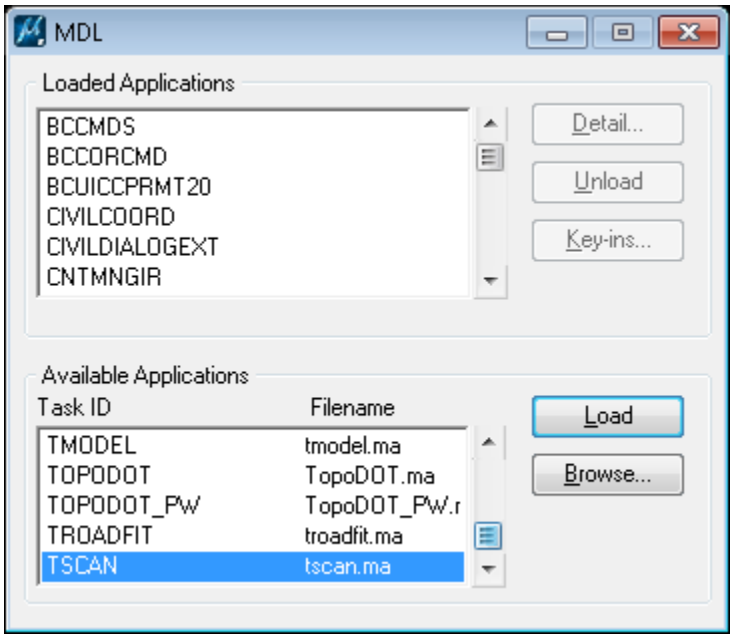
Starting Terrascan

Terrascan is an MDL application that is run within Microstation (not a tool but more of a plugin which uses the Microstation interface)

To start the MDL, select Utilities → MDL Applications

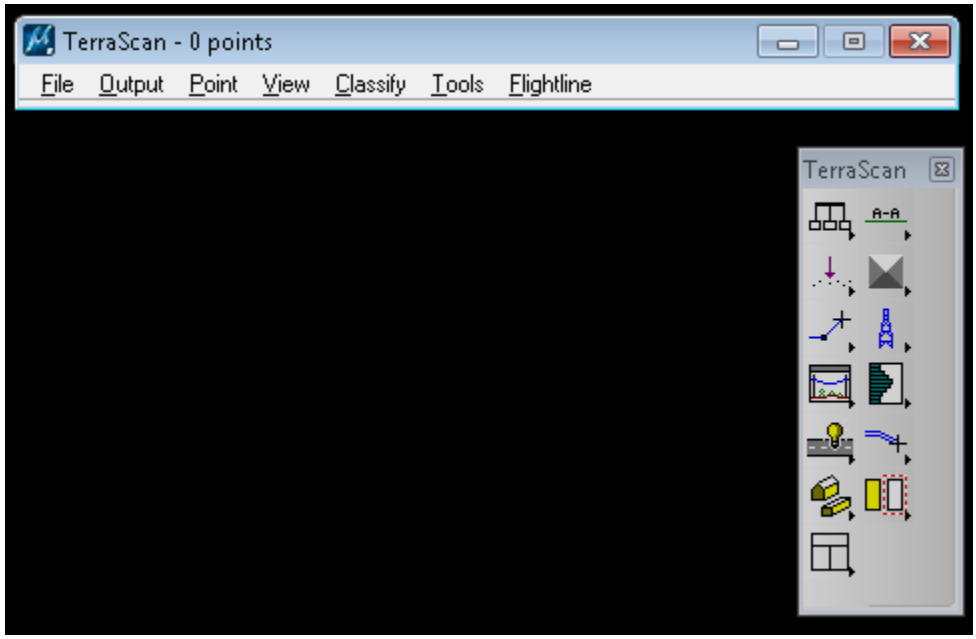


An MDL dialog box will appear (depicted on the next page) under the Available Applications section select the TSCAN task ID and then click load. (The MDL window can be closed)



TerraScan opens two windows (shown below).

Main Toolbar



Read Points in TerraScan

Using the Main toolbar (depicted in the above photo in red) select File → Read Points

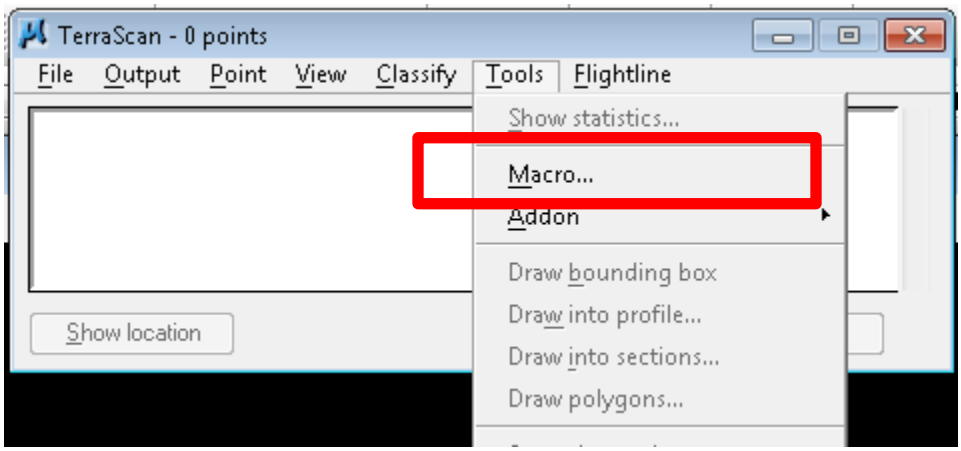
Select the file you wish to load and open (a .las file is the standard and best format to use)

Load Points dialog box will appear (Use the default settings) and click OK to load the data.

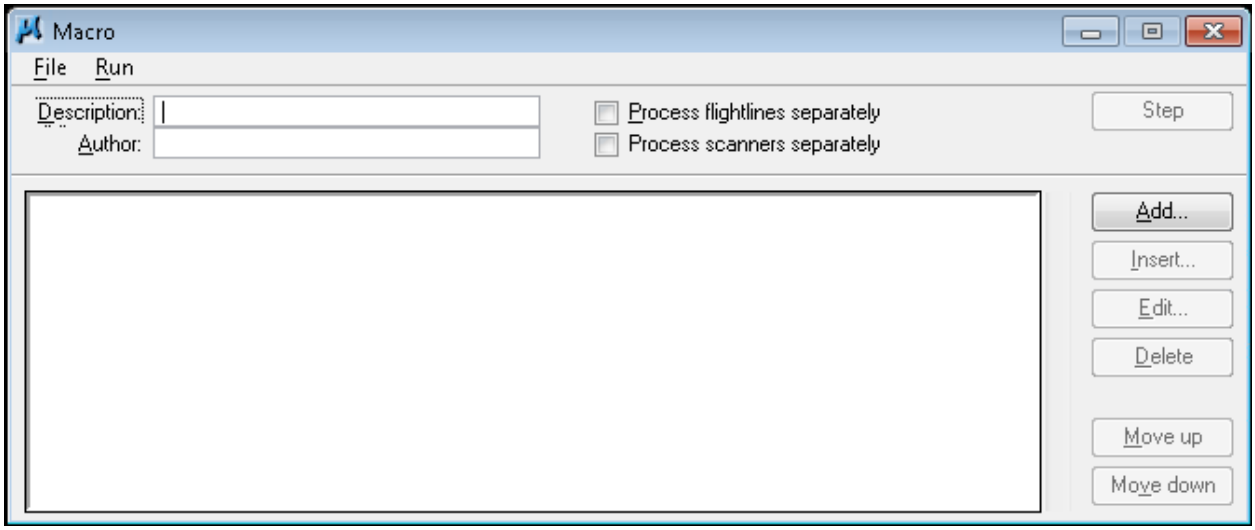
Running a Macro for Auto-Classification

To run a macro on the loaded points you can utilize pre-written macros or create your own using the macro tool.

First, in the main toolbar, select tools→ macro



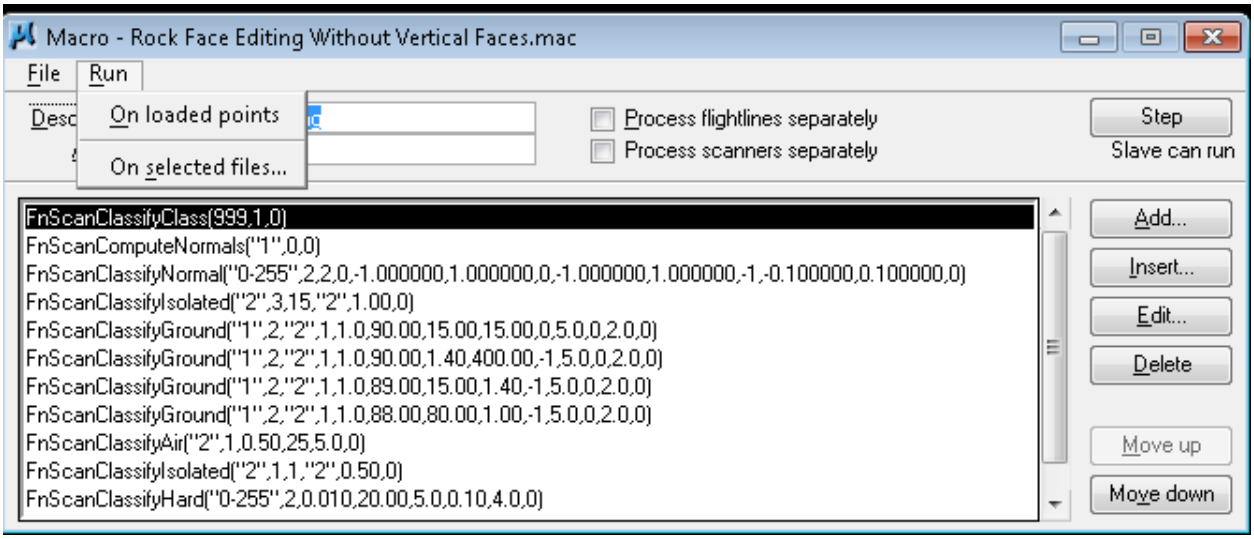
The macro window will open.



Under File select Open.

Browse to the .mac macro file that is pre-written or skip these last steps and add steps using the macro tool.

To run the file on the entire loaded point cloud select Run→ On loaded points



The macro will run through the entire script.

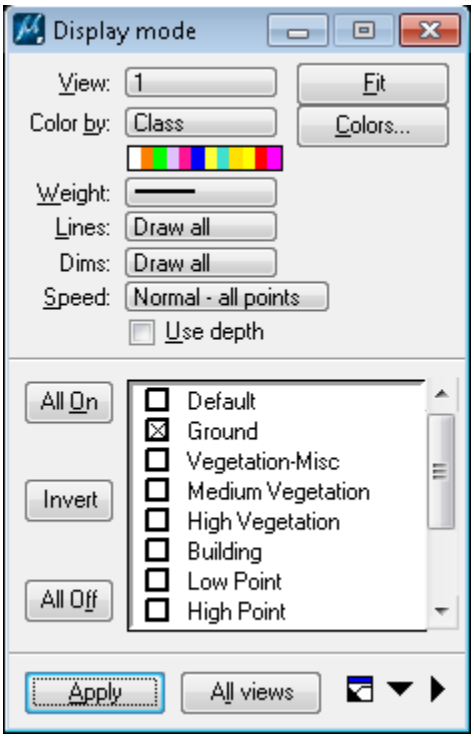
*To step through the script, use the Step button located in the top right of the macro tool.

Manual Editing

Within TerraScan there are a few tools that allow the user to manually edit the point cloud for those points which might not be classified correctly or the user wishes to not include in the ground (or tin).

To visualize the data in the most efficient manner the display mode tool can be used.

Under the view tab in the main toolbar select display mode



Chose the view you are displaying the points in (or all views) and under speed select normal all points (to ensure all the points are displayed)

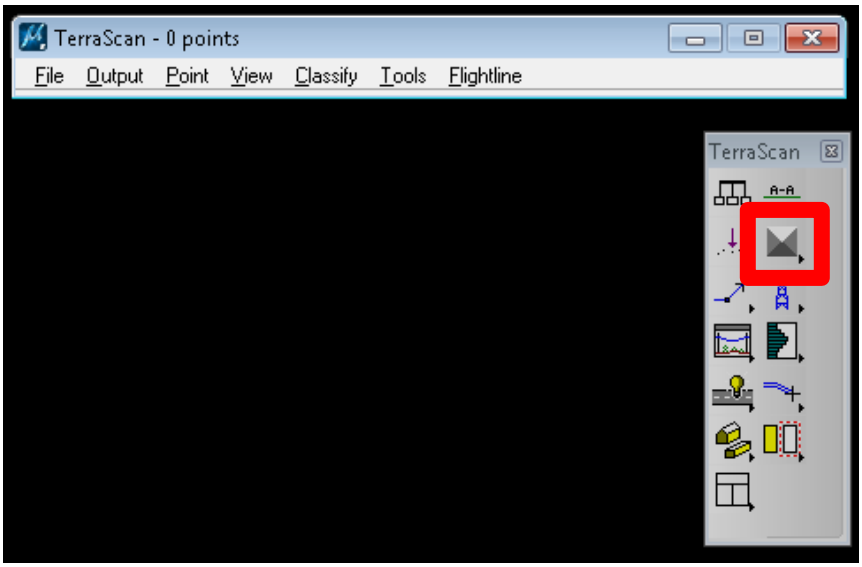
Turn off all the classes and select the ground class. (This will only display the ground class so when the user is using the brush tool it will appear as if the points are getting deleted but instead they are being reclassified and placed in a different class).

TIP* Under the view tab in the main toolbar there is a fit view tool this will fit all the active points to the view the user clicks inside. (As opposed to using the global fit which will fit everything in the drawing)

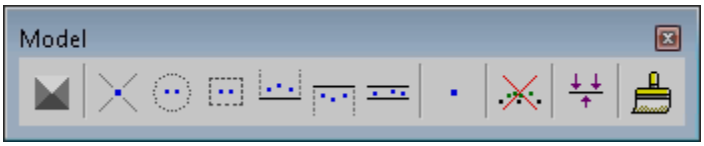
Classify Brush

Next open the classify using brush tool

To open this tool the model toolbar needs to be opened. (TerraScan tool bar boxed in red on the next page.)

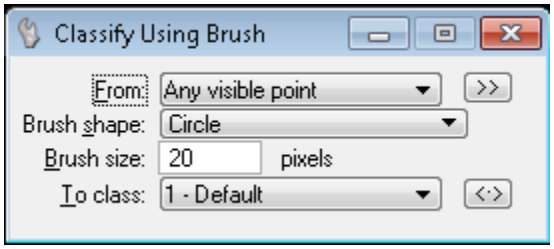


To undock this toolbar to easily expand click and drag the toolbar out of the docked position.



The picture above is the toolbar undocked.

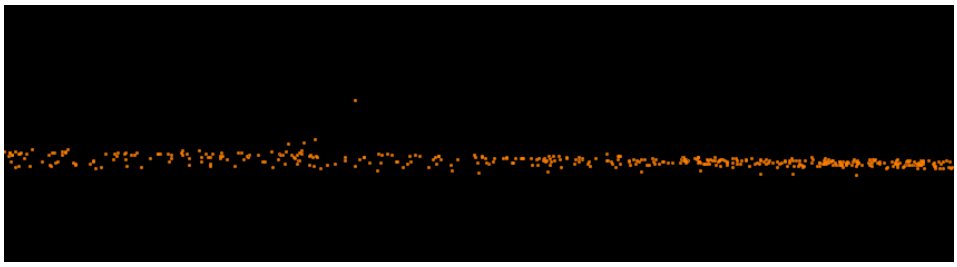
The “classify using brush” tool  is opened by clicking on the icon.



Note* a tip for ease of use is to increase the brush size to 20 pixels.

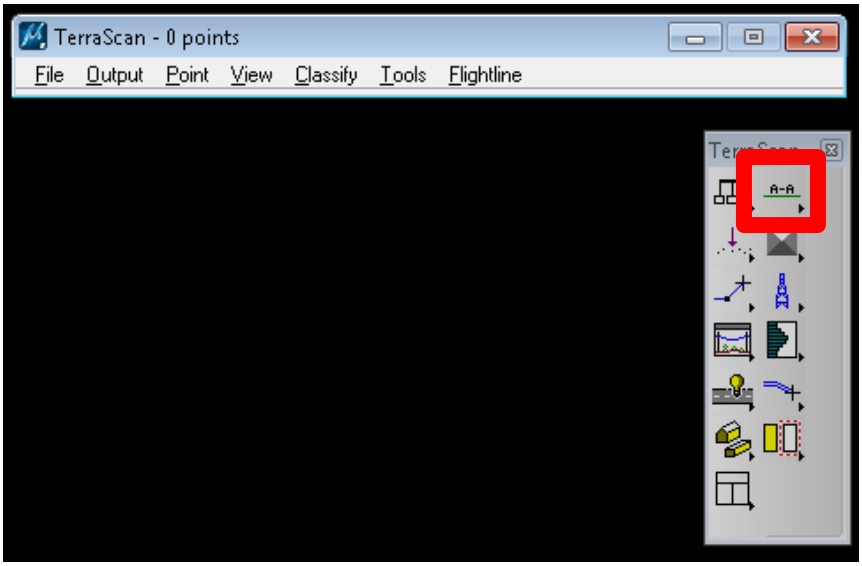
To use the tool the user can click to turn the brush on and click to turn off or click and hold to keep the brush on until the left mouse button is released and simply “brush” over points that need reclassified.

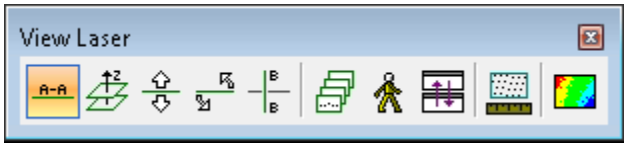
A best practice is to rotate the point cloud onto one side to look at the surface level to find the points that the algorithm misclassified.



Vertical Cross Section Tool

The vertical cross section tool is very useful for editing large detailed point clouds in conjunction with the brush editing tool. To open this tool, utilize the TerraScan Toolbar and click and hold the A-A section button (to dock, click and drag to the location you would like to dock).



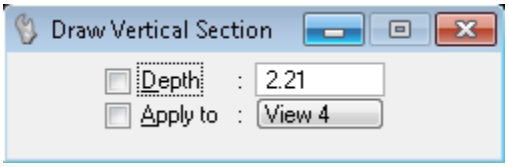


To “cut” a cross-section select the A-A tool

Data click once to start the cutting plane and data click a second time to define the width of the plane.

A third data click depending on the settings will give the section’s depth.

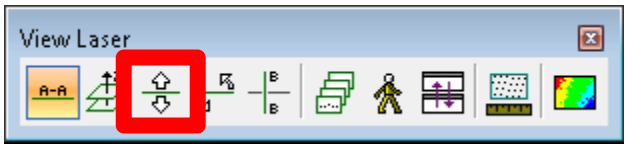
TIP* A max section of 10 ft. is recommended (bridges require a smaller depth).



Using the Draw Vertical Section dialog box a depth can be prepopulated (if the check box is on the user does not need to define a depth.

A fourth (or third if the depth box is checked on) data click will pick which Microstation window the section will be drawn in.

The user can now utilize the classify tools to classify the section. Once completed the user can "walk" through the data using the move section tool (boxed in red) and repeat. A left click advances the section while a right click will move to the previous section.

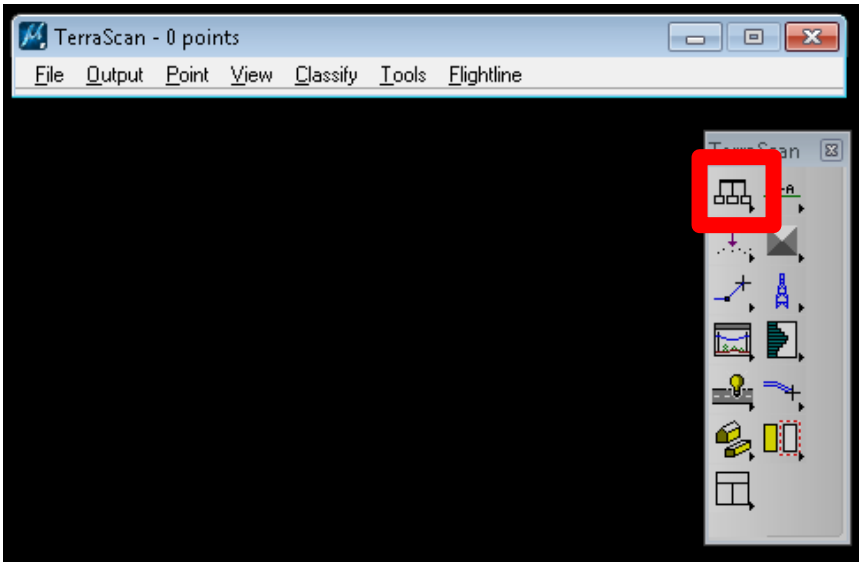


Scaling Point Clouds

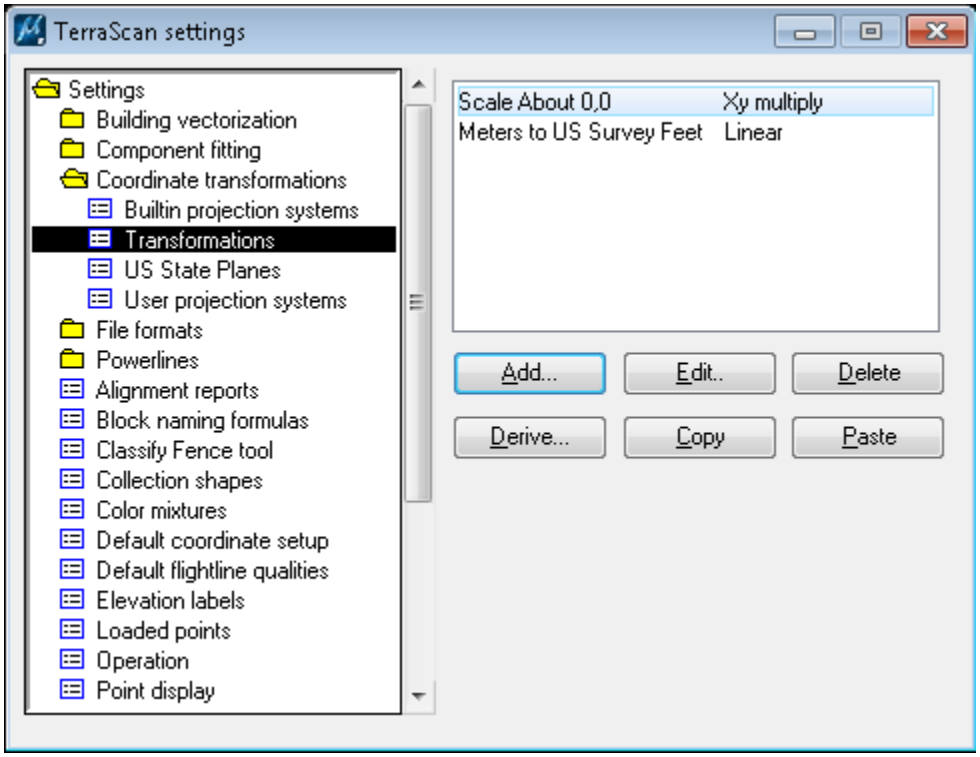
This guide will cover two different methods for scaling point clouds. One is for grid to ground applications and the other is for unit changes (e.g. meters to feet).

Custom User Defined Transformations

Under the TerraScan toolbar click on the TerraScan settings button.



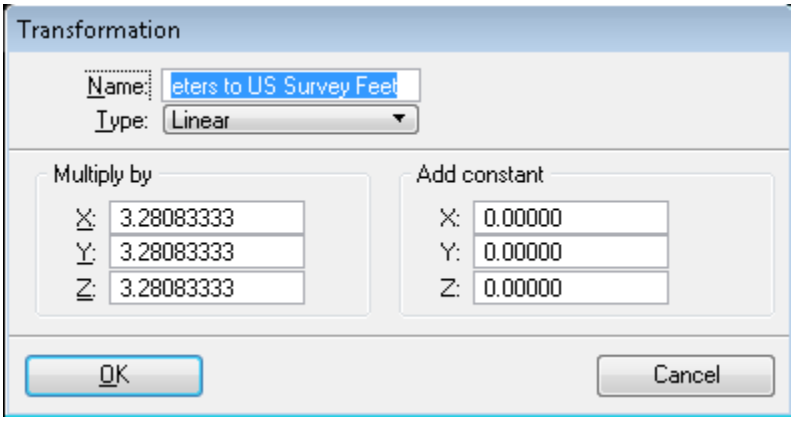
A dialog box will open.



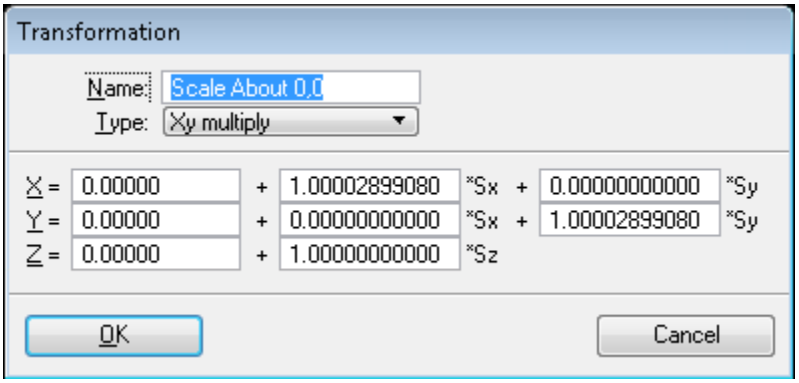
Under the Coordinate Transformations select Transformations.

For some users, a couple custom transformations may be registered already.

If the unit change transformation is not there (e.g. Meters to US Survey Feet) simply click the add button and select the linear transformation type. Then Click OK to save.



If the Scale about 0,0 is not there simply click add and select the Xy Multiply transformation. Then click OK to save.

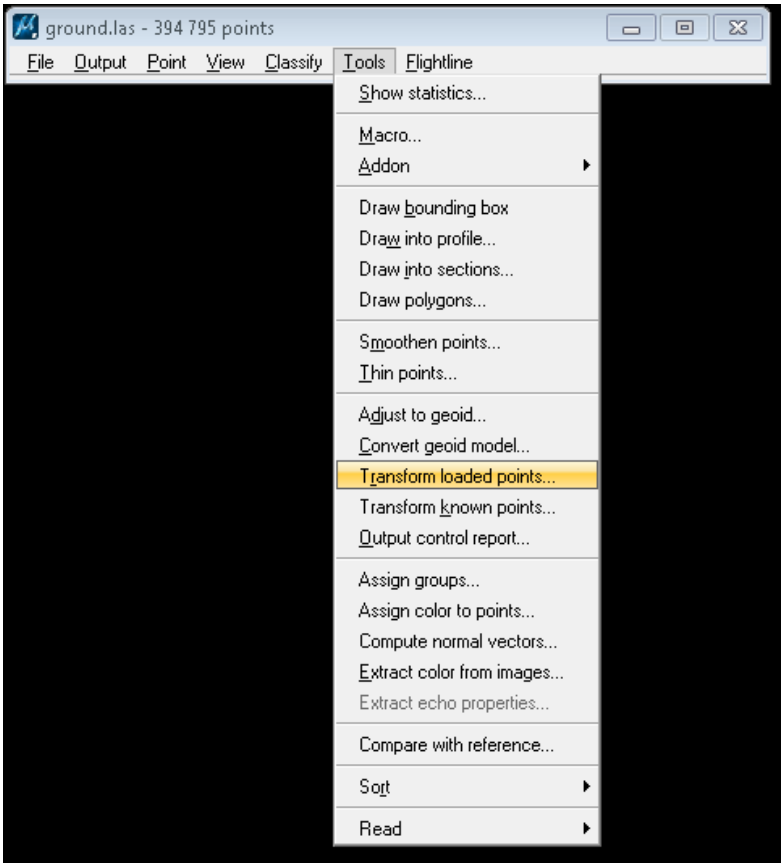


*Please note that every project you are trying to scale will need to have this updated and saved with the appropriate scale factor.

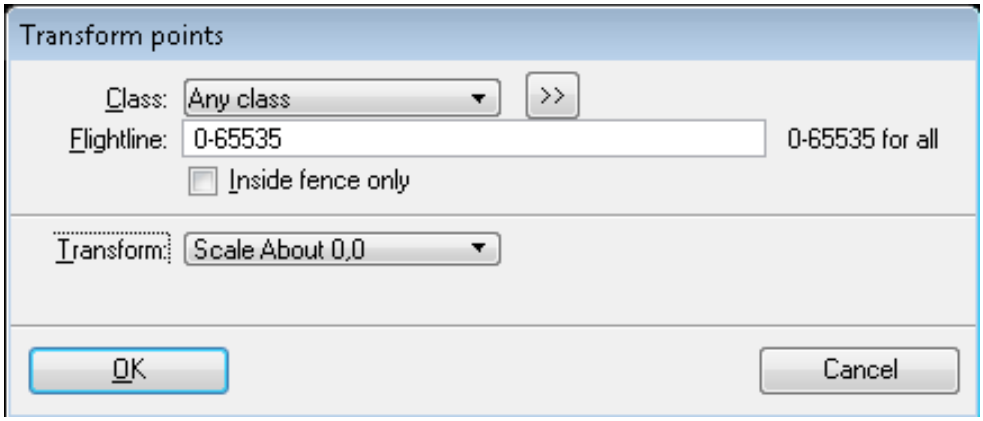
Applying Transformations

This workflow will be very helpful when exporting point cloud data out of Leica Infinity (even if the project is in feet the data will export in meters) or when a point cloud needs scaled to ground after the fact.

With a loaded point cloud, under the main toolbar select Tools→ Transform Loaded Points.



The Transform Points toolbar will open.



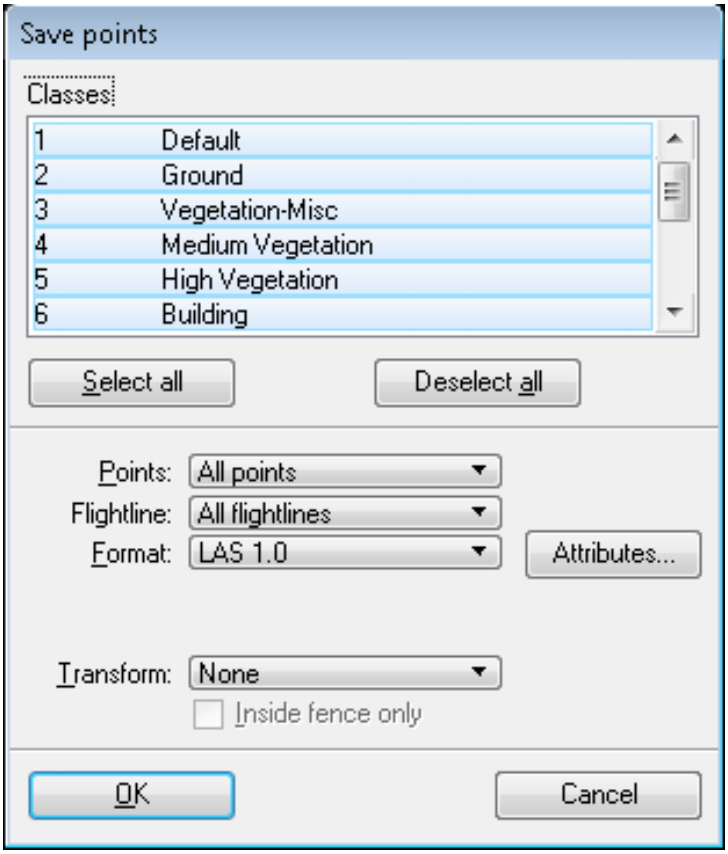
From the drop-down menu select the transformation you would like to apply. Then Select OK.

Exporting Processed Point Clouds

When exporting (saving) points out of Terrascan, users can save by class (e.g. ground points) or by region created by a fence (or a combination of both).

To save your data click File → Save As in the main Terrascan toolbar.

The Save Points dialog box will open.



Select the classes you wish to export and if using a fence click on the Inside fence only selection box. Click OK.

A standard windows file save dialog will open. Browse and name the file then hit save.

A dialog will appear showing how many points were written to the file.

Please close out of the remote desktop connection when finished with this step.

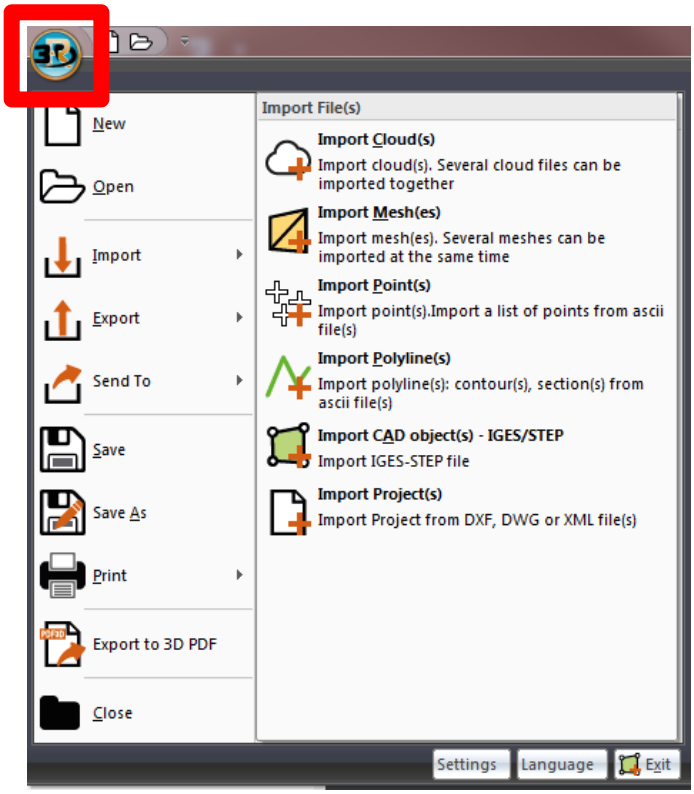
3DReshaper

3DReshaper is a powerful tool for creating mesh surfaces, volume calculations, and change detection models from point clouds.

This section of the guide assumes the data has been edited and exported out of Terrascan or a similar product in a .las format.

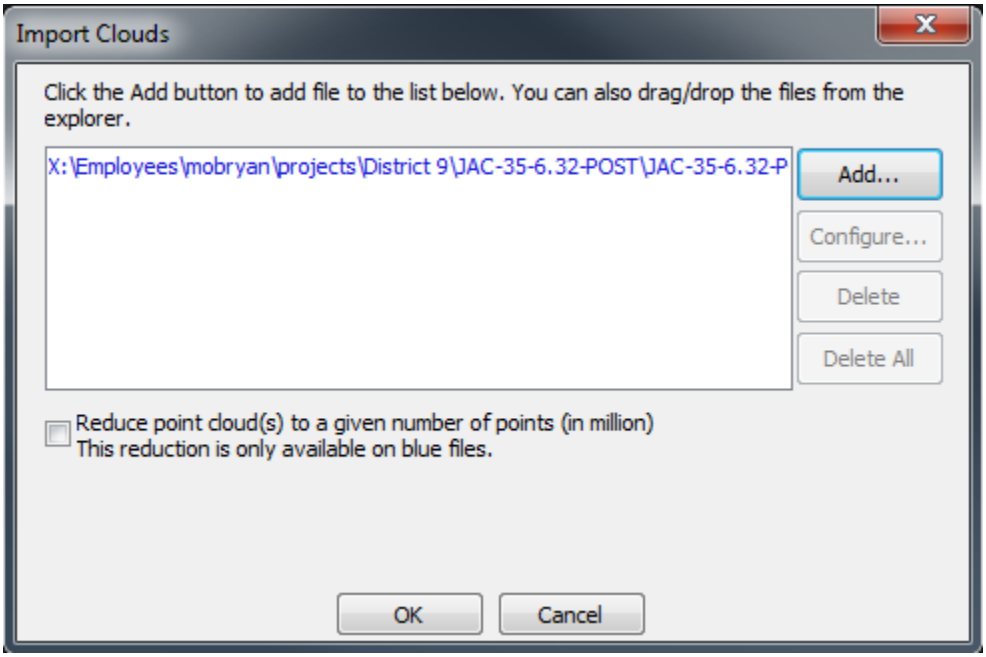
Loading Point Clouds Into 3DReshaper

After opening 3DReshaper select the file menu button located in the top left.



Go to Import → Import Clouds and select the point clouds you would like to import.

The Import Clouds dialog box will appear.

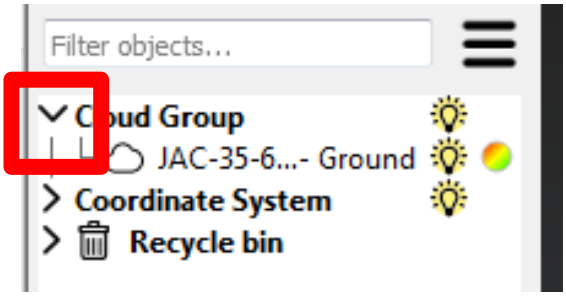


Click ADD to browse to the point cloud you would like to import. Uncheck the Reduce point cloud(s) to a given number of points box to retain the detail of scan.

Click OK to begin the Import.

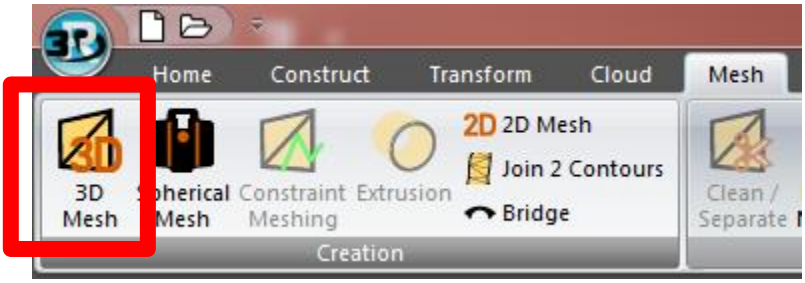
Creating a Mesh

Once the desired point cloud data is imported expand the cloud group in the 3DReshaper Tree (typically docked on the left side on the window).

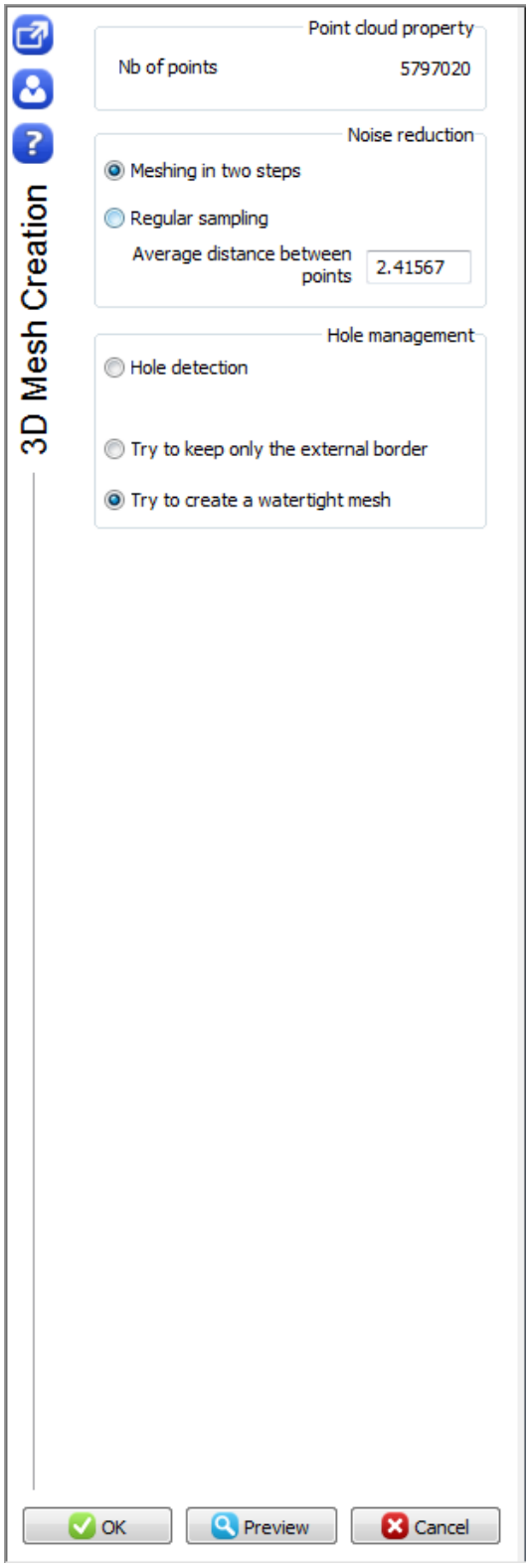


Select the file or files to be meshed

Under the Mesh tab located in the tool ribbon select the 3D Mesh tool.



Once selected the 3DReshaper Tree will turn into the 3D Mesh Creation toolbar on the left side of the window.



Toggle the Noise reduction parameters to Meshing in two steps and (depending on the spacing of the scan) set the average distance between points (the default typically works well).

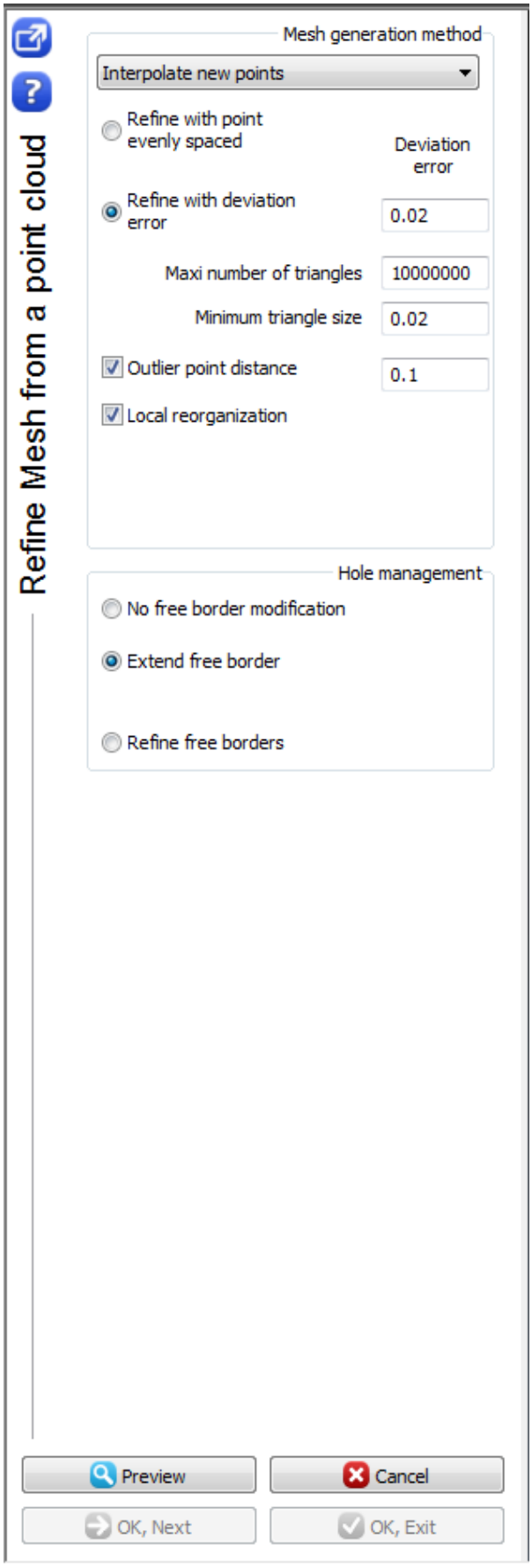
In the Hole management section please toggle the create watertight mesh on.

* Certain applications of meshing will require the hole detection method be used (e.g. beam modeling). If not the mesh will enclose itself in areas that do not accurately represent the beam.

** It is best practice to utilize the Preview function then select OK to move onto the next meshing step.

Select OK

The 3D Mesh Creation toolbar will turn into the Refine Mesh from a point cloud toolbar.



The second step of the mesh making process involves the refinement of the initial mesh created in the previous step.

In the Refine mesh from a point cloud dialog define the “refine with deviation error” (the smaller you go the more detailed the mesh will be but the file size will grow exponentially).

*A best practice would be around .02

For the Max number of triangles for large areas this number can be increased.

A minimum triangle size is driven by the scan density (generally set it smaller than the scan grid).

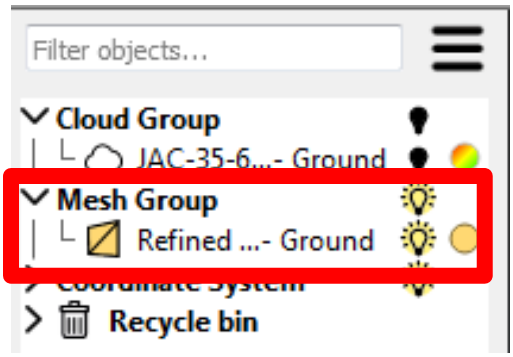
The defaults for the rest of the settings provide great results.

Select the preview button to check the settings.

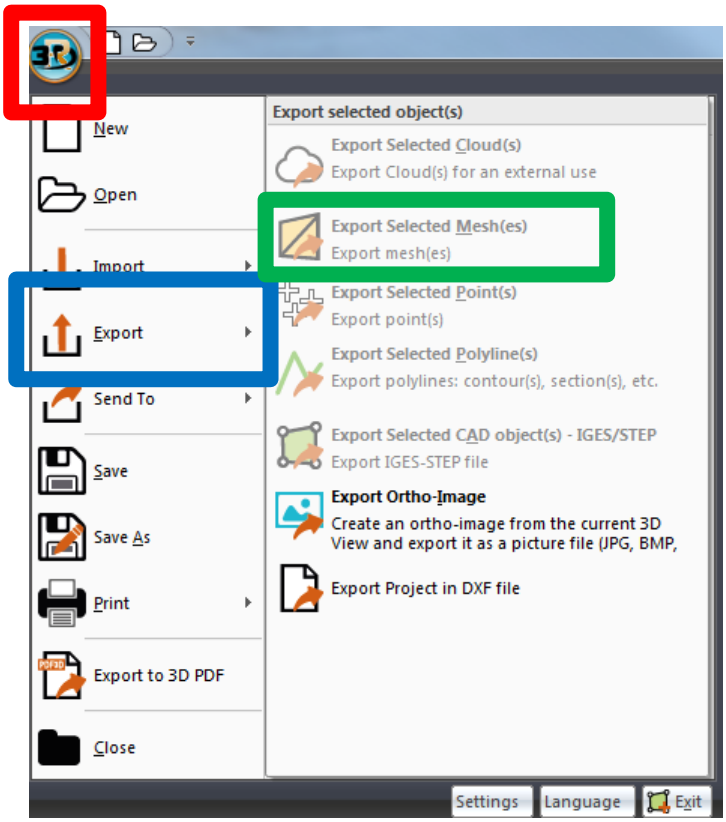
If the result is acceptable press the Ok, Exit button to finalize the mesh.

Exporting a Mesh

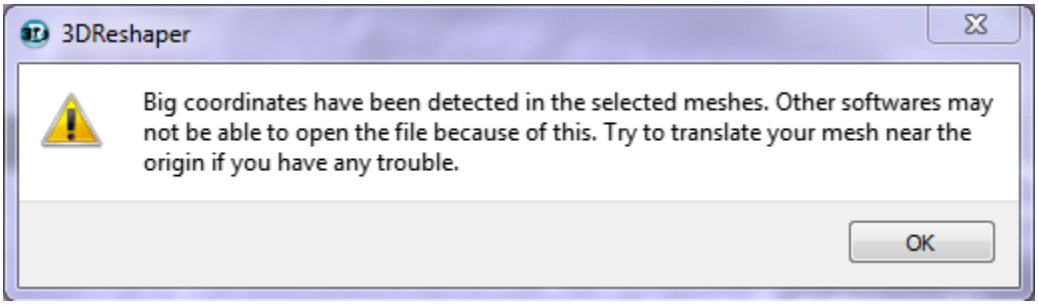
To export a mesh, expand the Mesh Group in the 3DReshaper Tree and select the mesh you wish to export.



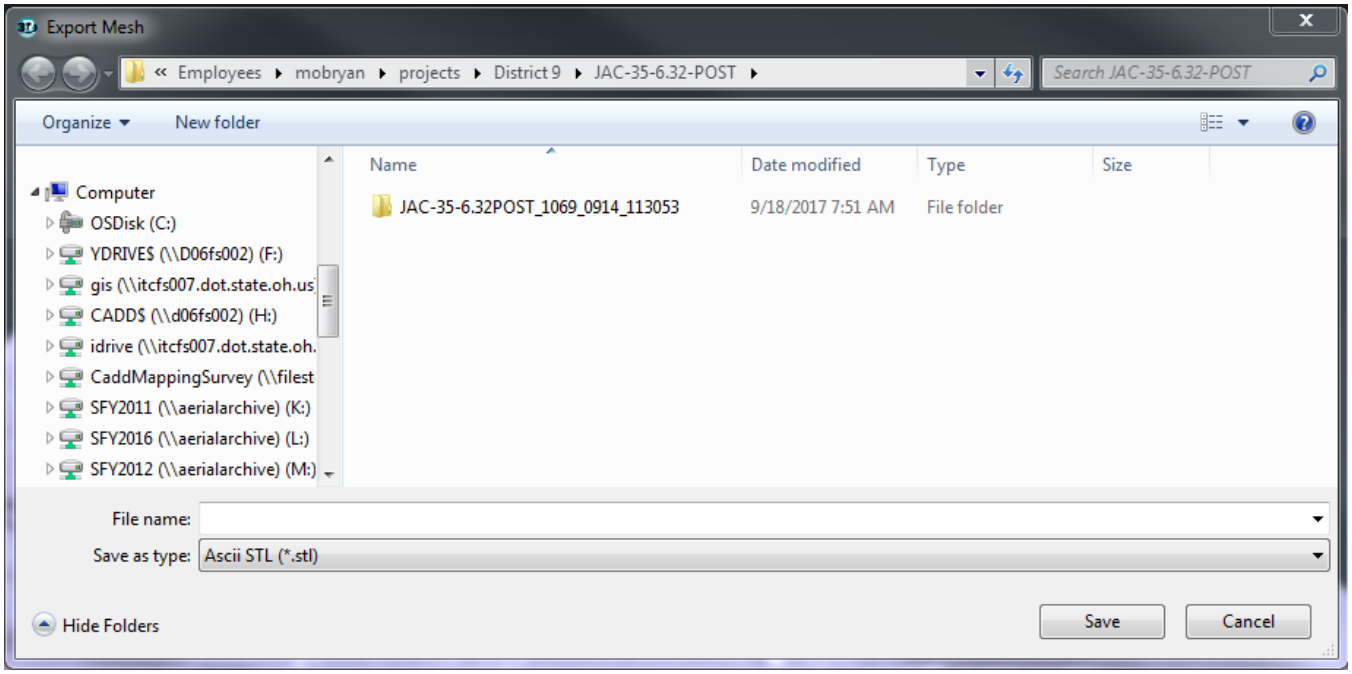
Once selected click on the file select button in the top left (red) of the window and browse to the export section (blue). Select the Export Selected Mesh(es) option.



If you are using State Plan coordinates you will receive a message warning about big coordinates. Press Ok to continue.



The Export Mesh dialog box will appear.



In the Export Mesh dialog box change the save as type to Ascii STL (*.stl). Browse to the location you would like to export and click Save.

This file format can be opened in Microstation and other software packages.

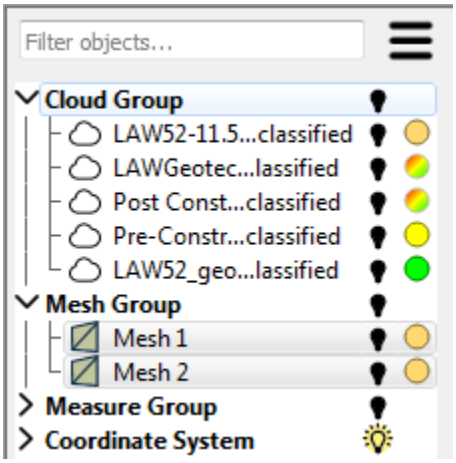
Change Detection Models

Change detection models are a great way to visually display and document changes that have happened to an area (structure or terrain).

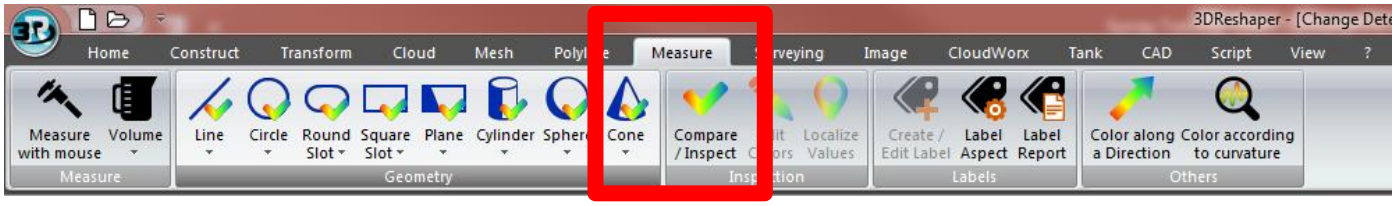
This modeling (basically colorizing) can be ran on both point clouds and meshes, utilizing the same workflow.

To begin load in the point clouds or meshes you would like to compare.

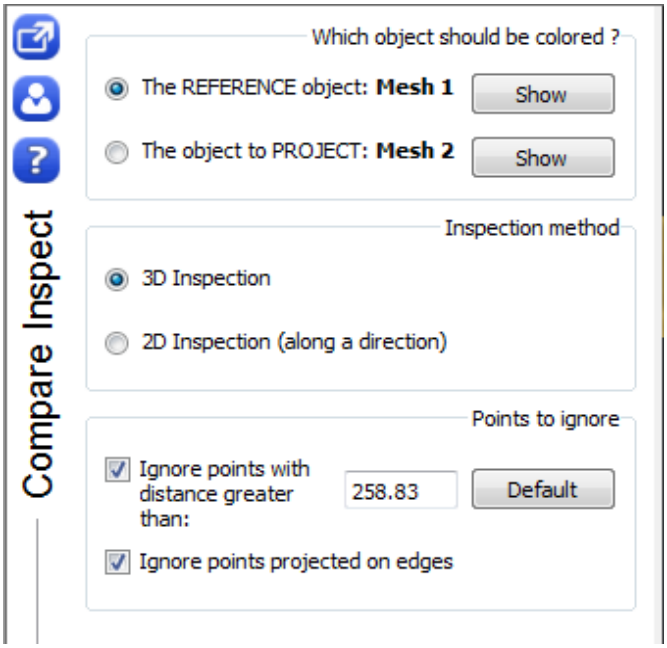
In the 3DReshaper tree (located on the left side of the window select the meshes or point clouds in the order you would like to compare them.



Once selected under the Measure tab at the top select the Compare/ Inspect tool from the Inspection group.



The 3DReshaper tree will turn into the Compare Inspect tool on the left side of the window.



In the top section of the tool you can choose which point cloud (or surface) the colorization will occur on.

In the Inspection method please select the 3D Inspection method.

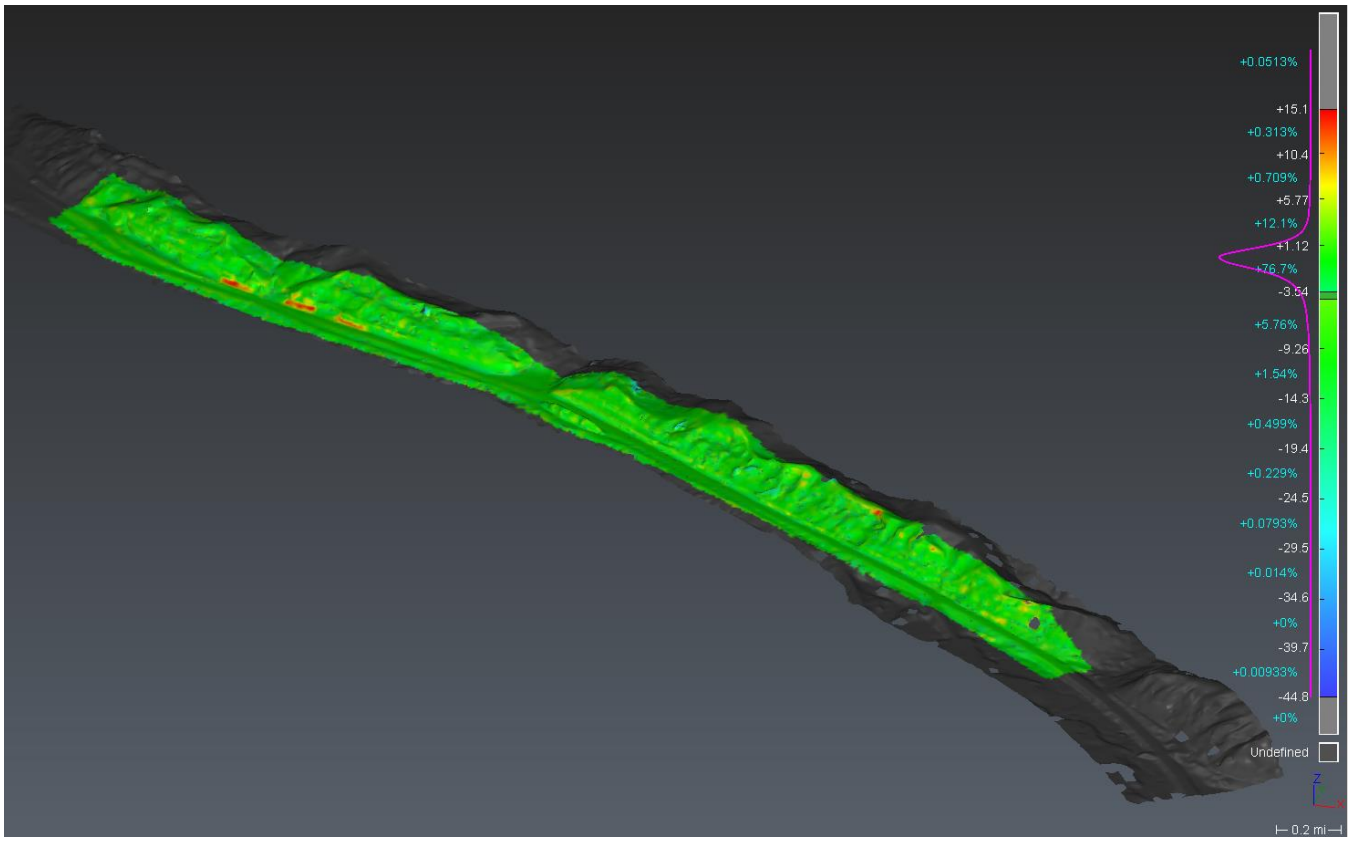
If you want to filter the points down further you can do that in the next section.

After the settings are assigned click the preview button. This will display the results on the cloud/surface in a preliminary visualization.

Once this is completed the colors can be edited (if you are looking for certain changes or emphasize certain features).

Click OK to finalize the change detection.

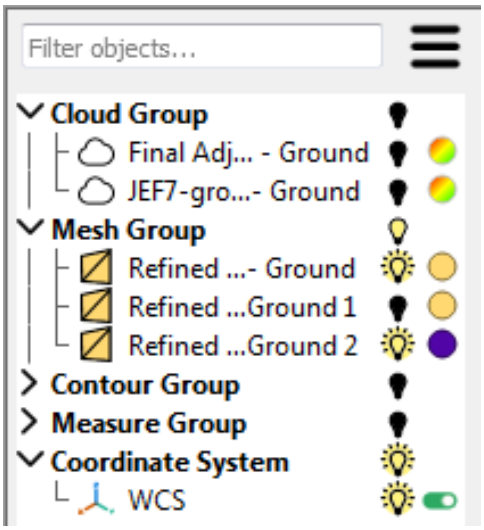
Below is an example of a mesh generated aerial LiDAR change detection for reference.



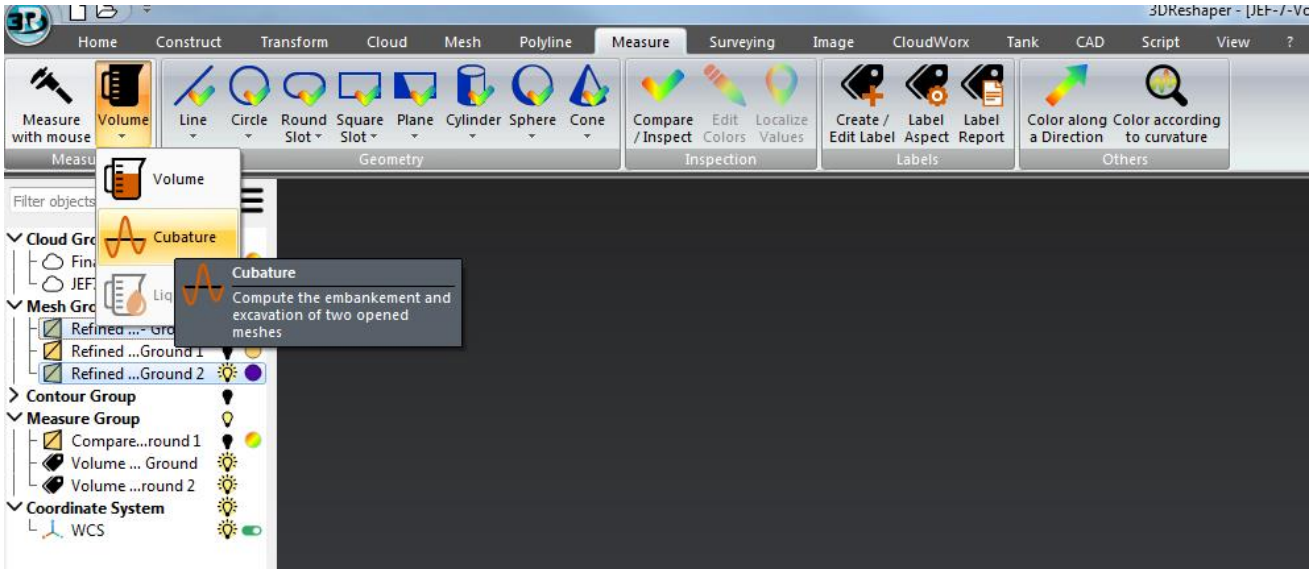
Volume Calculations

To do volume calculations, the user will need to have generated a pre-and post construction mesh surface of the area in question. (The mesh surfaces don't necessarily need to be created from point clouds (e.g. a mesh generated from design))

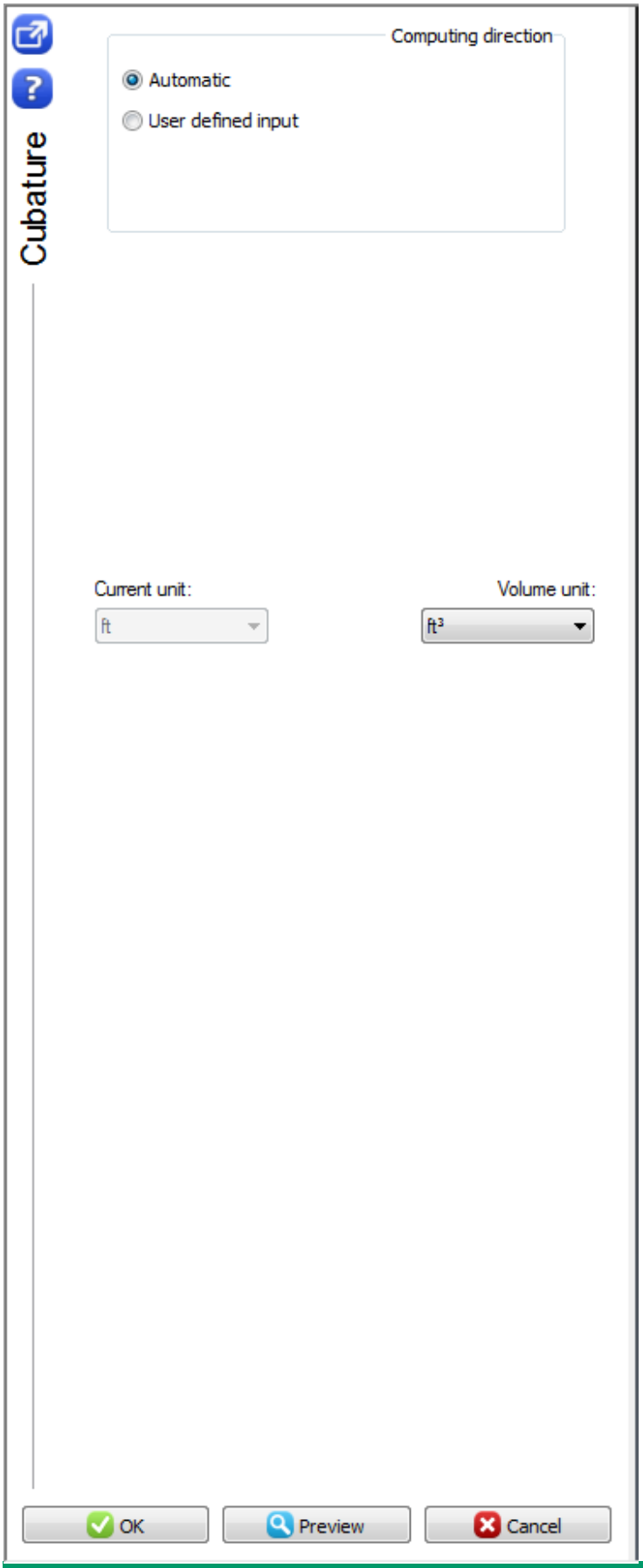
With the mesh surfaces imported or generated in a 3DReshaper file, select the meshes that encompass a volume from the 3DReshaper tree.



Under the Measure tab at the top of the main window select the Volume tool under measure then select the Cubature tool. This tool will compute the embankment and excavation between two opened meshes



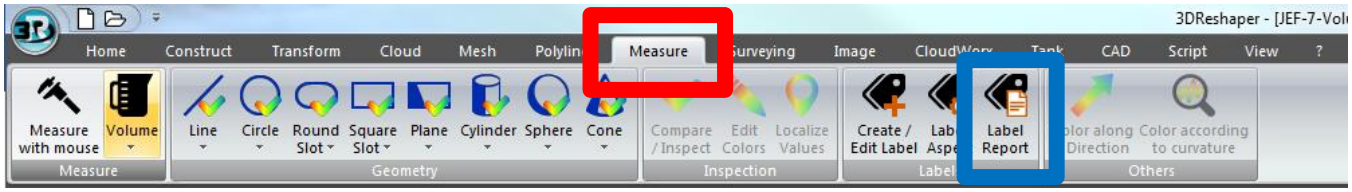
Once selected the Cubature dialog will replace the 3DReshaper tree on the left side of the window.



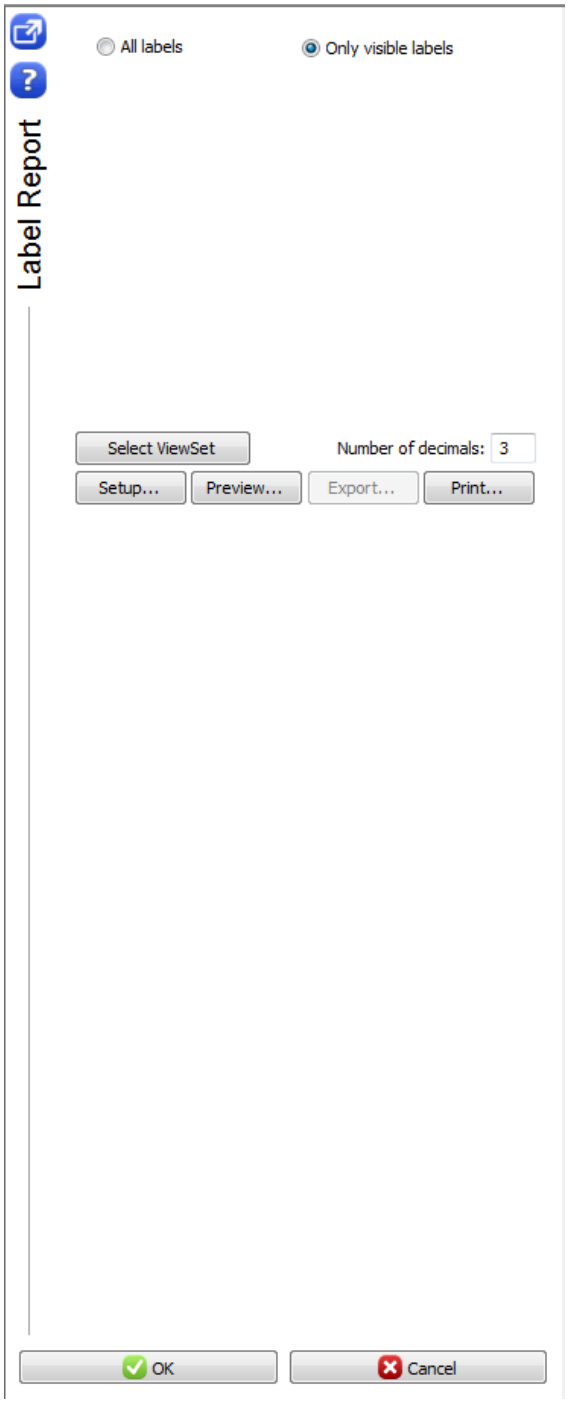
The tool is very simple. Select the Automatic computing direction and the units the volume will read out as.

Select either Preview to check your settings or OK to Finalize them.

To generate a report of the calculated volumes select the Measure tab (boxed in red) at the top of the window and under the label section select label report (boxed in blue).



Once selected the Label Report dialog will appear over the 3DReshaper tree.



Check on either all labels or only visible layers depending on if you've created additional labels.

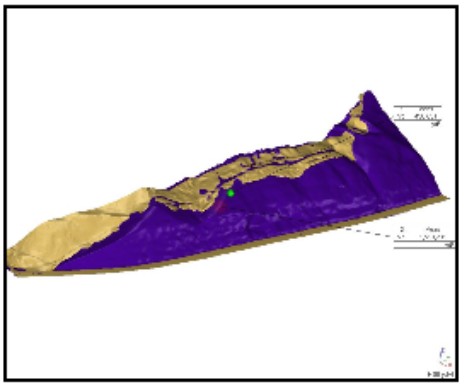
*Tip use the Preview to see if the formatting matches your desired settings.

When finished Print to pdf (or hard copy) and select OK to return out of the tool.

3DRESHAPER®
The 3D scanner software

HEXAGON

3DReshaper
www.3dreshaper.com
support@3dreshaper.com



Customer	**ODOT**	Product name	**Volumetrics**
Drawing number		Controller name	**CADD and Mapping Services**

Label N° 1 ; Volume above Refined from JEF7-ground-FINAL_2011 - Ground 2 and below Refined from Final Adjusted Combined Surface - Ground

	Meas	Excess
Vol	430651	☺

yd^3

Label N° 2 ; Volume above Refined from Final Adjusted Combined Surface - Ground and below Refined from JEF7-ground-FINAL_2011 - Ground 2

	Meas	Excess
Vol	1519201	☺

yd^3

Above is an example of a Report.

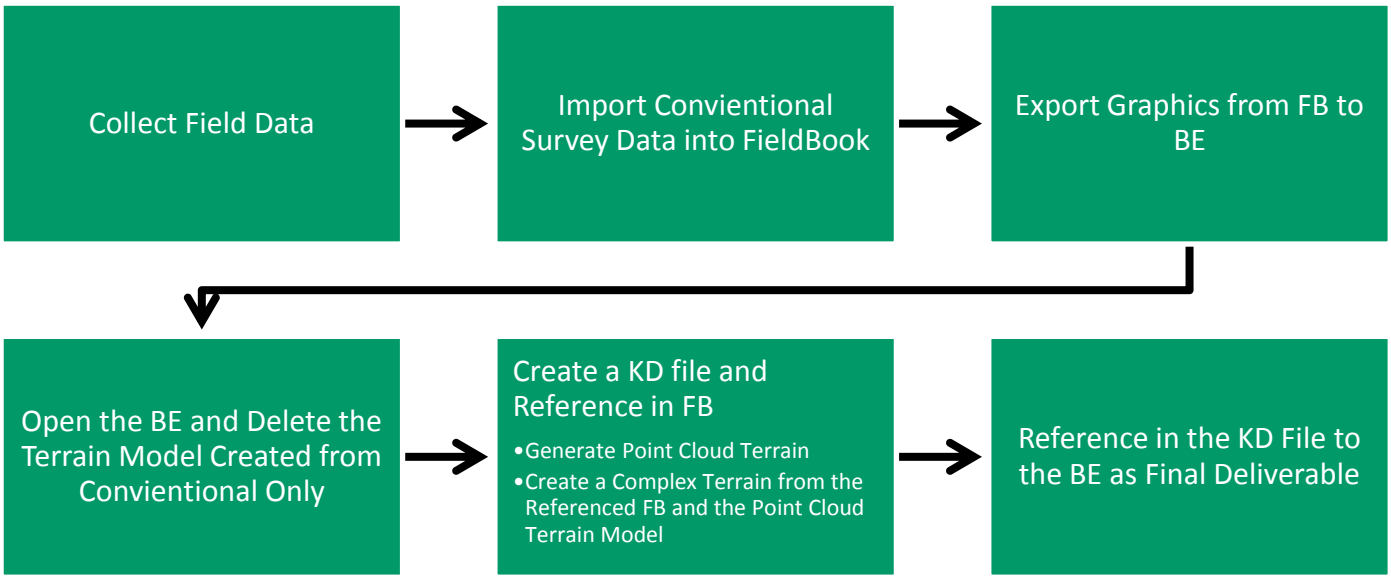
*Tip you can open a pdf in word to edit the formatting by right clicking on a saved pdf and selecting open with word. Word will then process the data into word elements that can be manipulated.

Microstation

Hybrid Terrain Generation

To generate a hybrid terrain consisting of conventional survey and point cloud data the following workflow should be followed.

Workflow

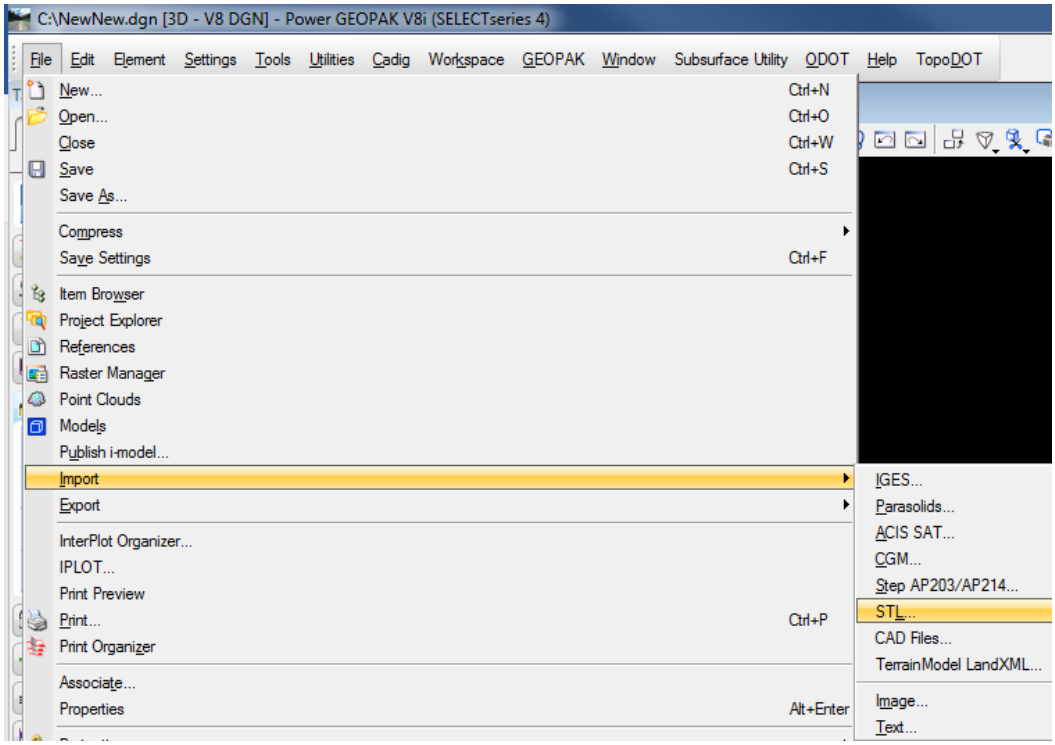


Importing Meshes

Microstation allows for the importing of certain mesh file types. One is the ascii .stl format generated in prior steps.

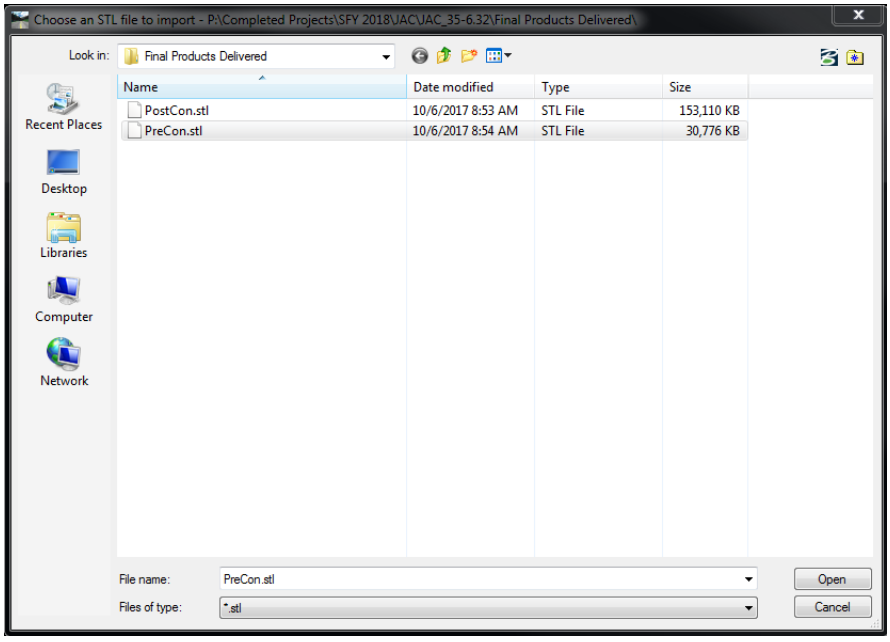
To import a mesh open a GeoPAK 3D file.

Under File select Import → STL

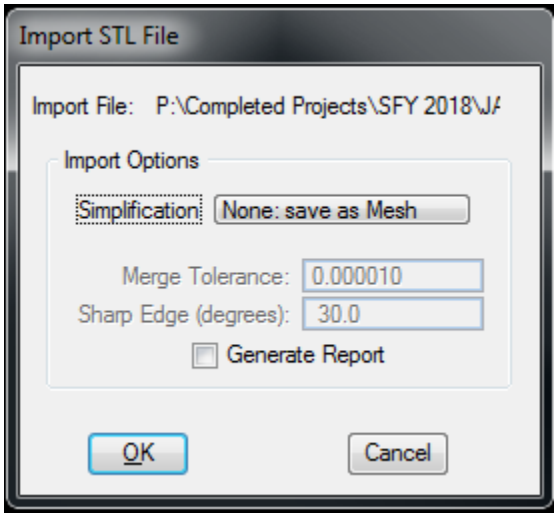


The Choose an STL file to import dialog box will open. Browse to the .stl files you would like to import.

*Tip to be able to turn the files on and off separately use different layers when importing.



Select Open. The Import STL File dialog box will open.



Utilize the defaults and select OK to import the file into the design file.

This file type (using open roads tools) can be cut by cross-sections just like a terrain model and compared to the design (for comparison after construction).

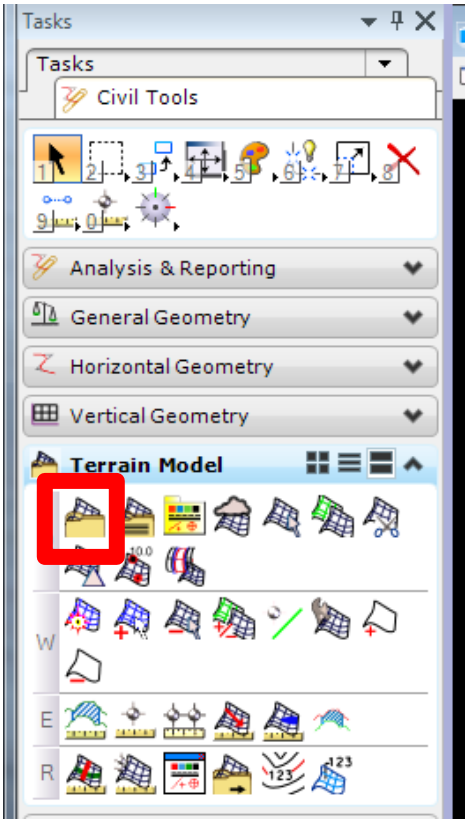
Generation of TIN from Point Cloud

**Completed using PowerGEOPAK

To quickly generate a tin from a point cloud we will be utilizing some built in Terrain Modeling Tools.

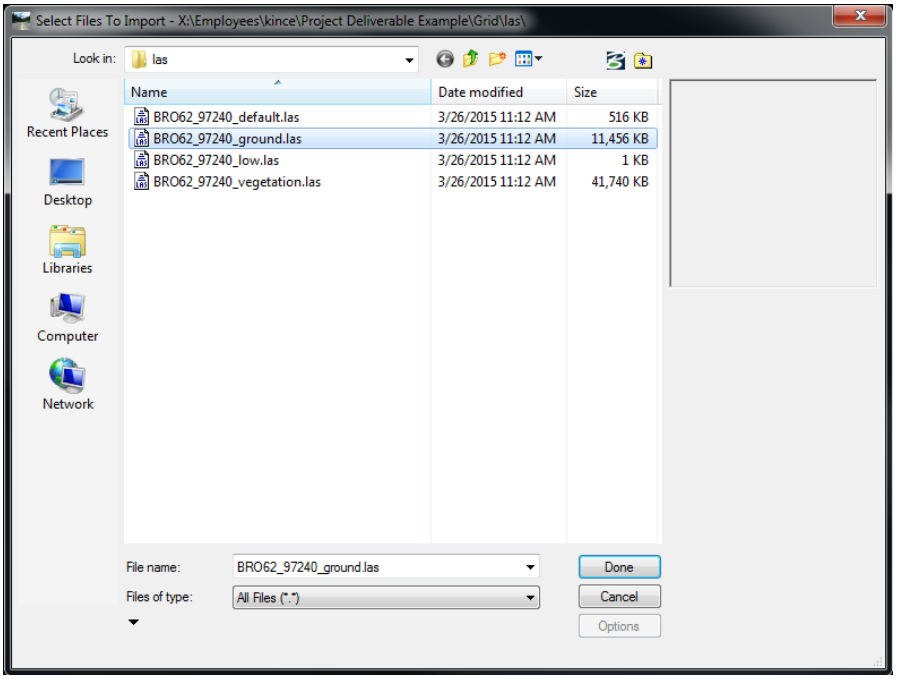
Under the Task Menu Civil Tools → Terrain Model → Create from File

The icon  is located on the next page highlighted in red



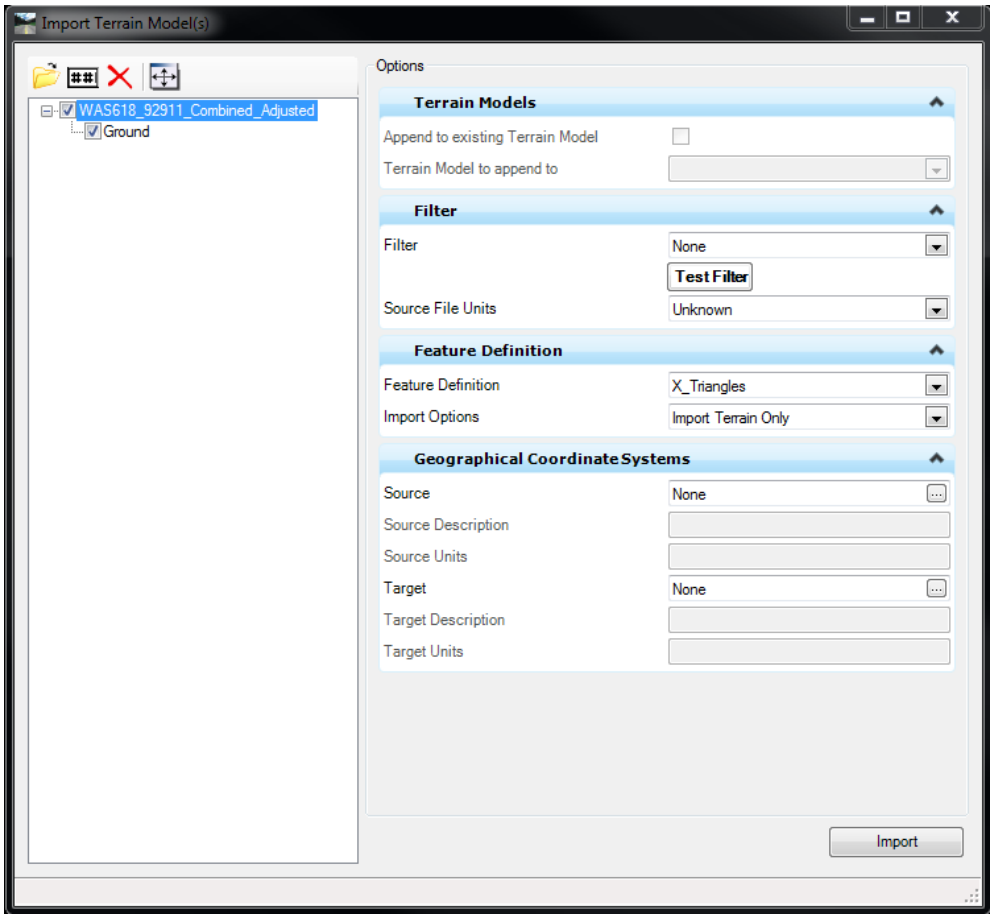
A “Select Files to Import” dialog box will appear.

Select the ground .las file created after editing. (The “Files of Type” filter might need to be changed to “All Files (*.*)”)



Click done when the correct files have been selected.

Now an Import Terrain Model(s) dialog box will appear. (Shown Below)



Utilizing the settings depicted on the previous page (and described below) Import the data into the design file.

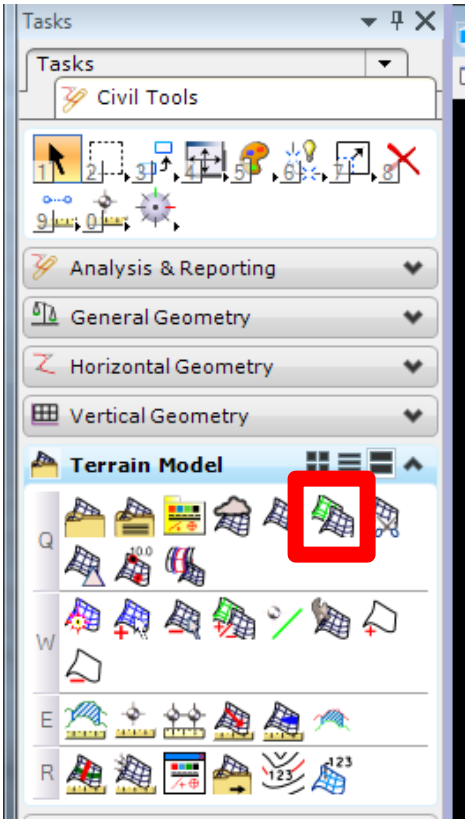
- Terrain Models Section
  - This section will be greyed out unless you are appending data to an already existing model.
- Filter Section
  - Filter
    - Use the Tin function here to allow the user to select the Z tolerance.
  - Z Tolerance
    - This function will filter out points in the terrain that do not add any additional elevation features to the limit specified by the user.
  - Reinsert Points
    - This function can be left off for typical tin generation.
  - Test Filter
    - This allows the user to see how many points will not be used in the creation of the tin (changed by the adjustment of the Z Tolerance).
  - Source File Units
    - All current CADD and Mapping Deliverables are in US Survey Feet.

- Feature Definition
  - Feature Definition
    - Depending on the deliverables the user can create several different layers that use the ODOT naming and visualization standards. For a typical tin use X_Trianlges. Then once imported the dialog box remains open to import contours or additional deliverables.
  - Import Options
    - For a standard deliverable Import Both
  - Include Spot Features
    - This function can be left off since the features haven't been generated yet.
- Geographical Coordinate Systems
  - This section does not need filled out due to the .las files containing Meta Data with this information (generally used for ASCII importing).

If not already done so please import the data

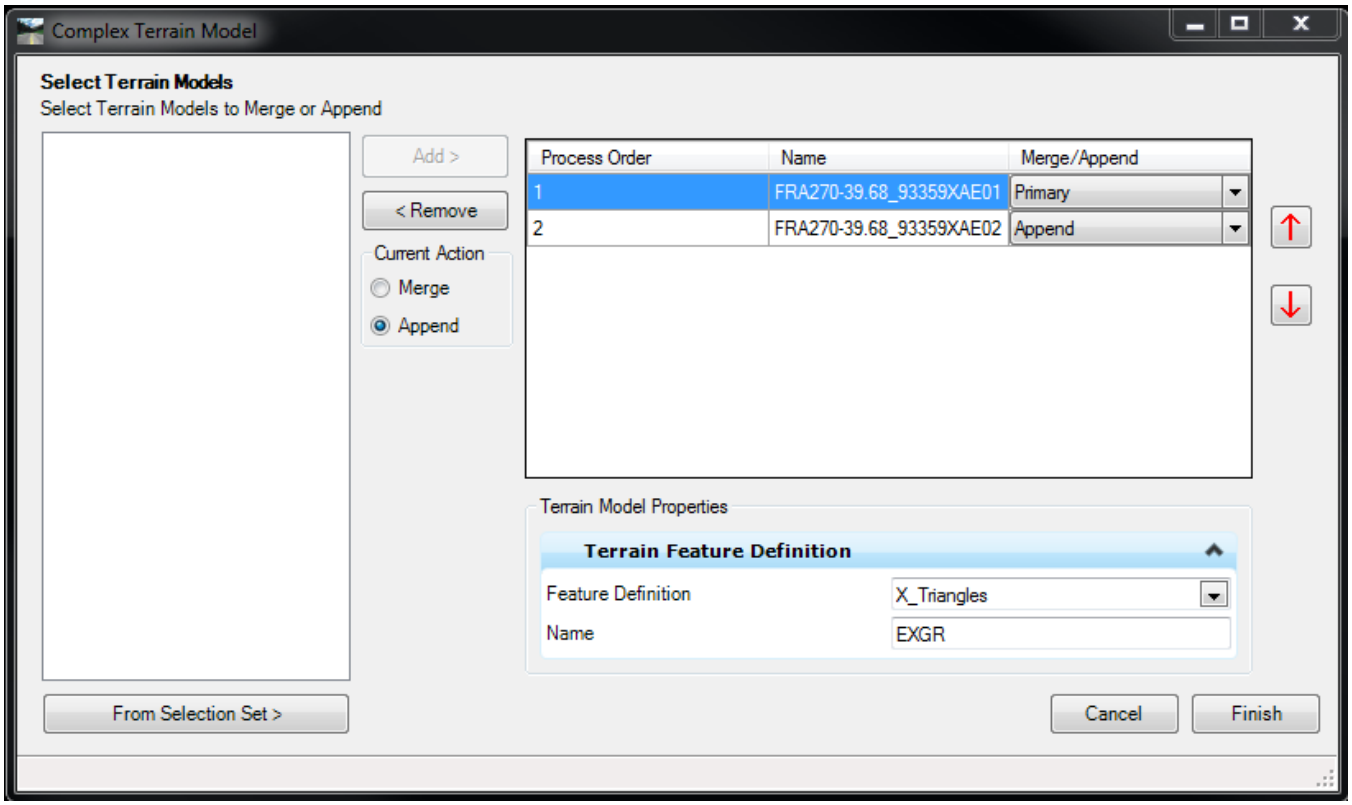
Creating a Complex Terrain Model

To create a complex Terrain Model, the user will need to either have the terrain models in the same dgn or attached as a reference (see Terrain Model Creation).



Select the Complex Terrain Model Tool from under the Civil Tools → Terrain Model toolbar (boxed in Red).

The Complex Terrain Model dialog box will appear.



From the left pane, select the files you wish to complex together. Then press Add.

Determine which file should be the primary and the appended file and select the Feature Definition from the drop-down menus.

Press Finish to finalize the terrain.

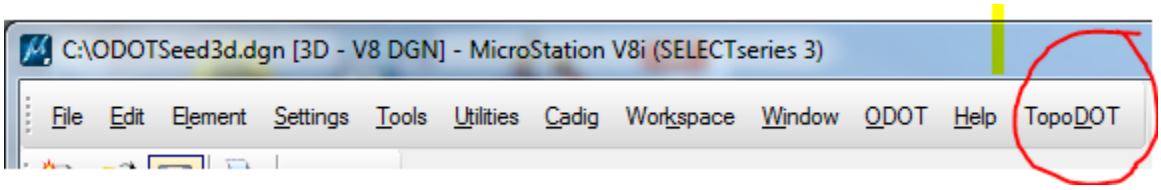
TopoDOT

Importing LAS files

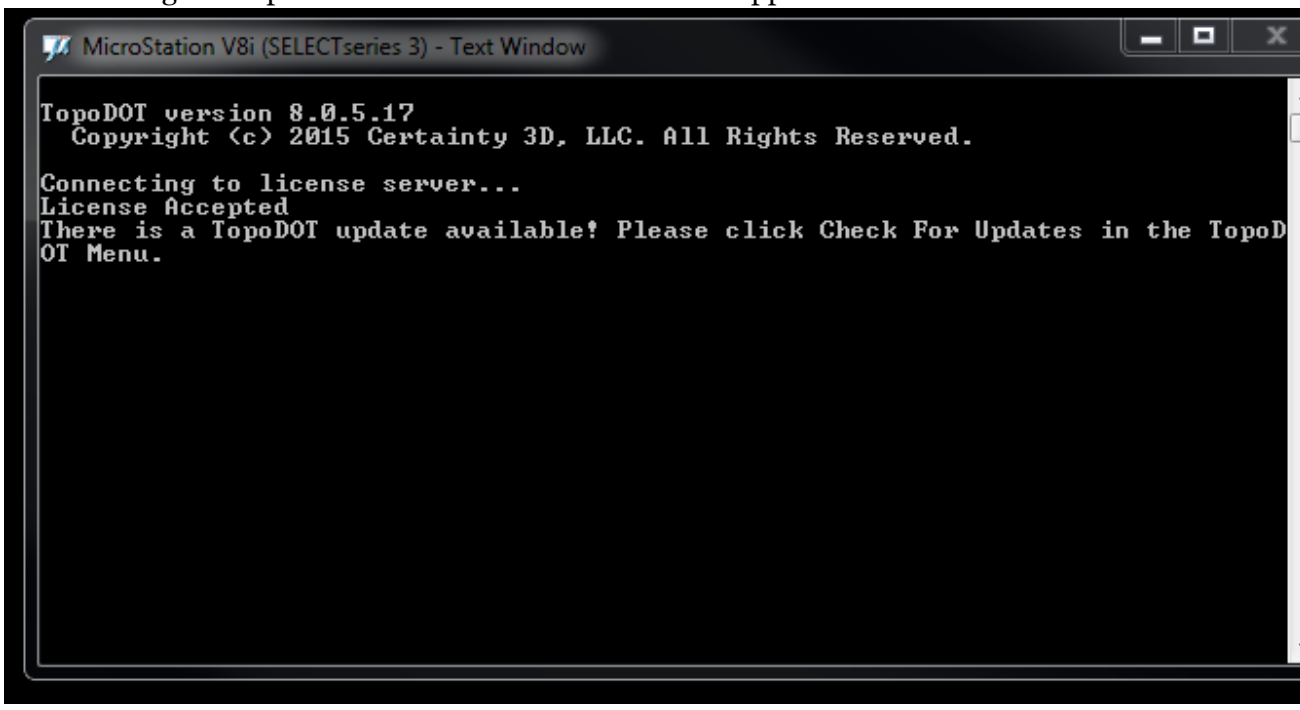
Step 1. Open Micro Station

Step.2 Select a 3D seed file.

Step 3. Click the TopoDOT button and select run TopoDOT

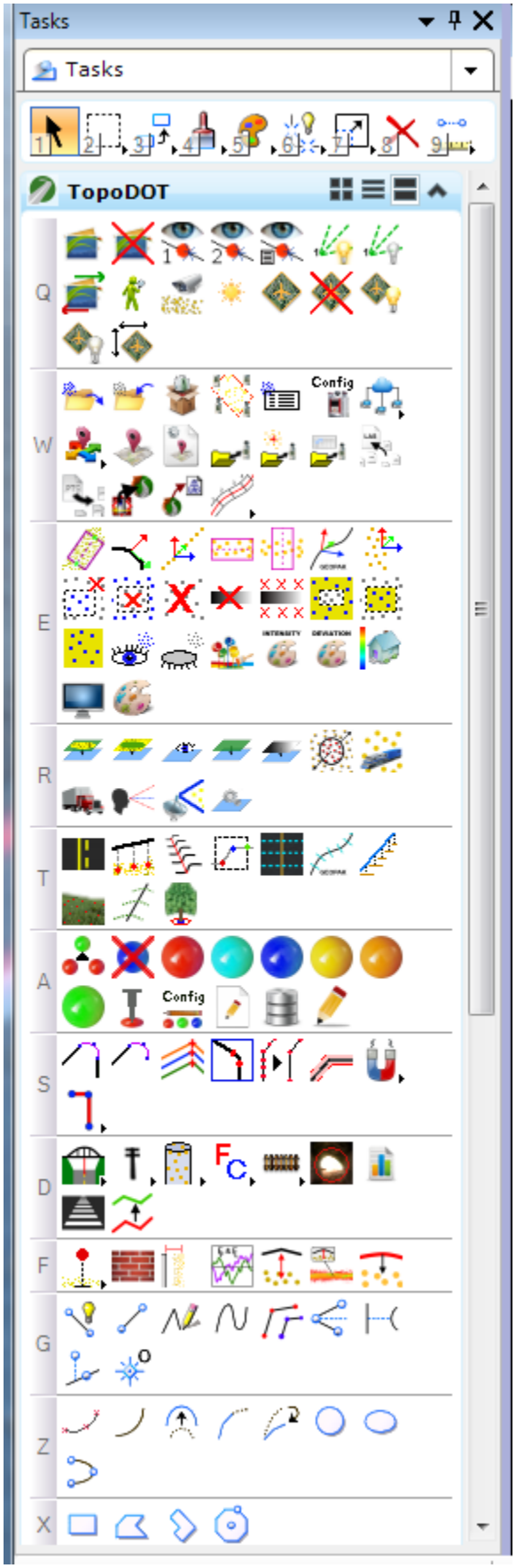


i. After clicking the TopoDOT button this text window will appear:

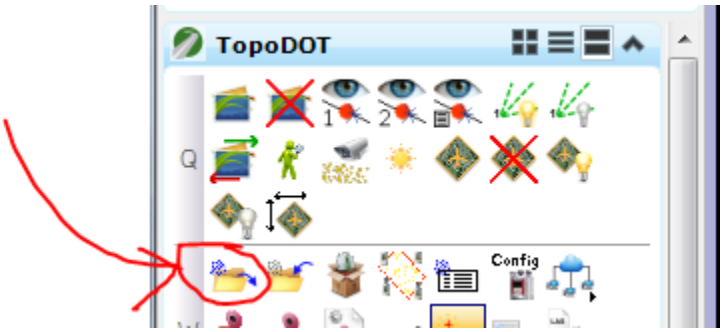


Don't exit out of this just let it run.

Step 4. Next select the TopoDOT tab under the task bar

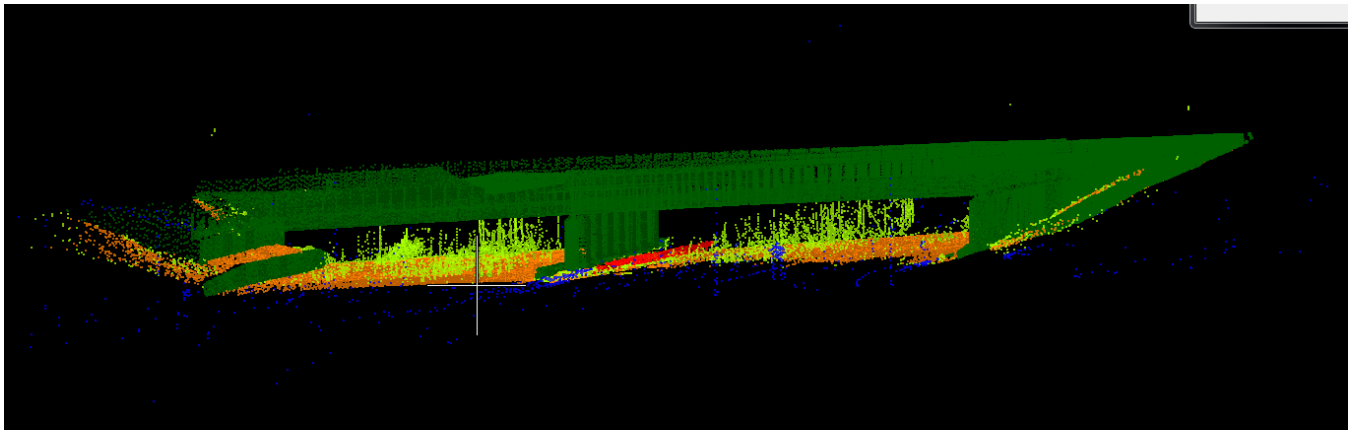

Tasks
Tasks
TopoDOT
Q
W
E
R
T
A
S
D
F
G
Z
X
Config
Config

Step 5. Select the Load Point Cloud From File tab



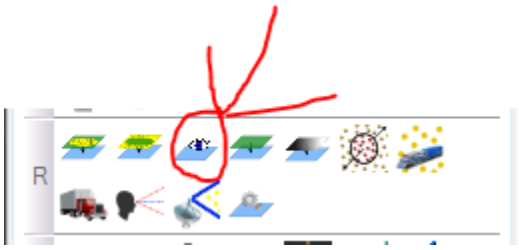
From this tab select the LAS file that you want

Step 6. Choose the display mode that you want your LAS file to be in (Classification, intensity, colored intensity etc.) This can be done using the View Settings tool located below.

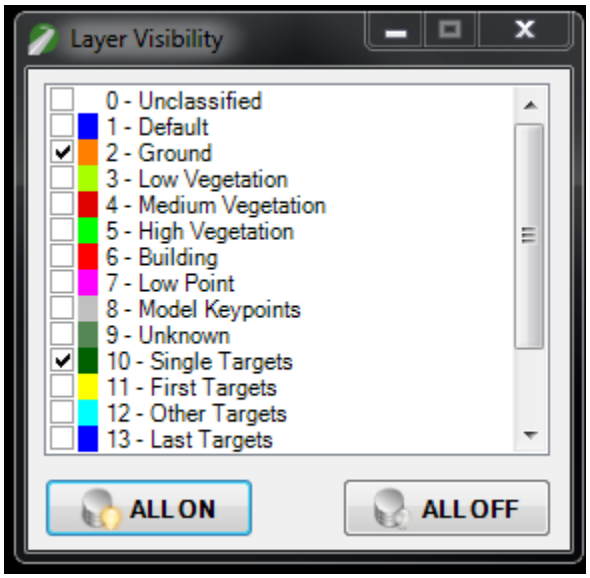


This is a Bridge LAS file that was scanned in Hamilton County on 275

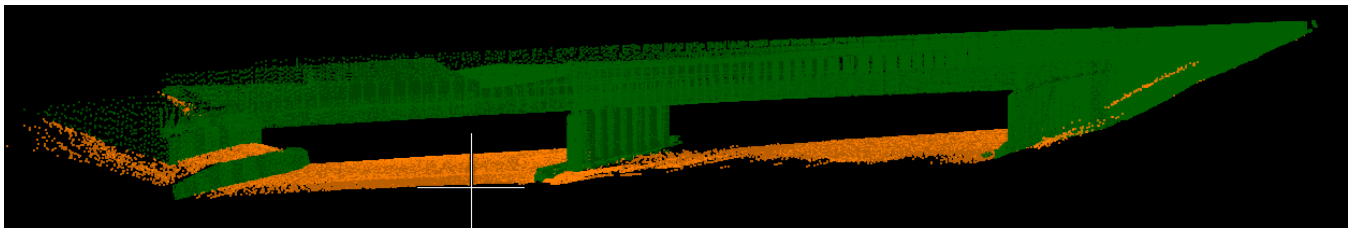
Step 7. Turn off the layers you don't need using the Layer Visibility tool



Once you click this tool this box will come up

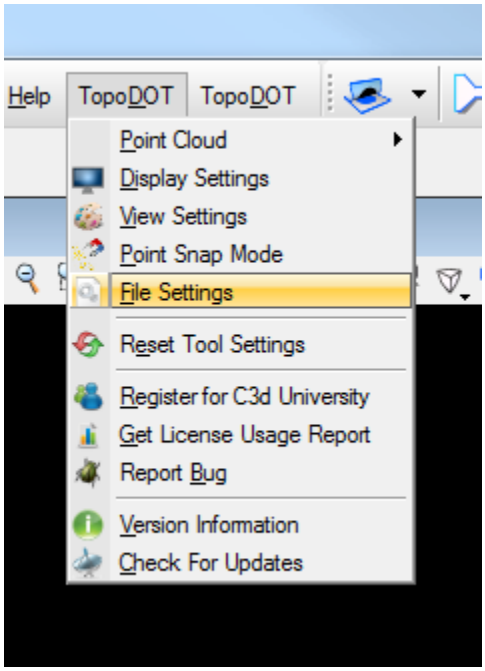


From this you can select the different class layers that you want to be turned on.



File Settings (Checking Units)

Click on TopoDOT tab and select File Settings



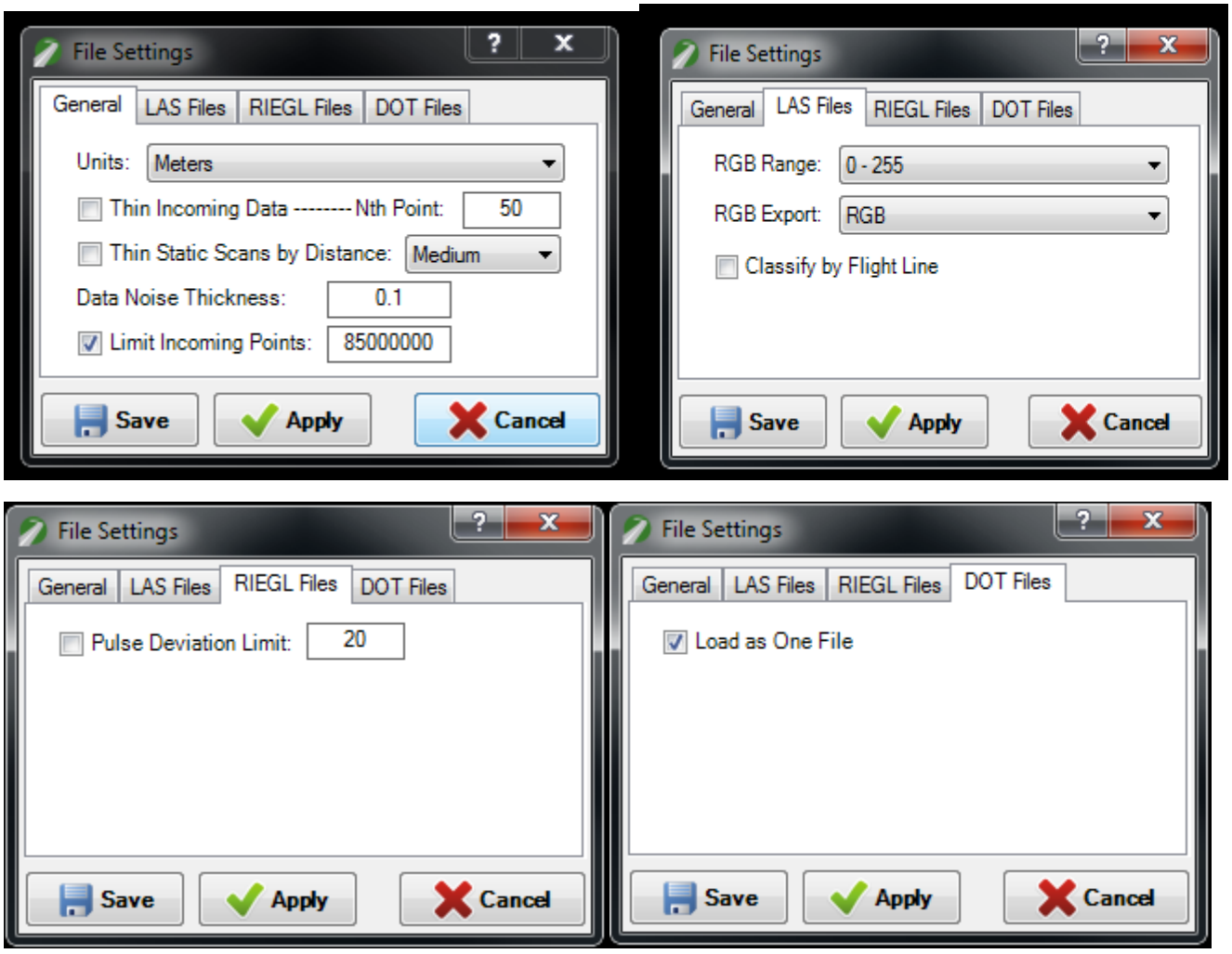
Purpose

Used to set units and other settings for the variety of Point Cloud files that will be read into memory. *NOTE: Design file units can be different then point cloud file units!*

- *Point Cloud files that do NOT have their internal units defined will require the 'Units' feature of this tool to be set correctly.*
- *If the point cloud files DO have their own internal units then the 'Units' feature of this tool will not affect the outcome of loading these files.*

Files WITHOUT internal units	*.LAS, *.LAZ, *.T3D, *.XYZ, *.ASCII, *.TXT, *.PTS, etc.
Files WITH internal units	*.DOT, *.RXP, *.E57, *.FLS , *.POD, *.3DD, *.CL3/CLR, *.BIN, *.ZFS, etc.

- *If a scaling issue happened it happened prior to the *.DOT creation when working with the 'Files WITHOUT internal units'. In this case, you must revert to the original raw files and set the units correctly before moving forward, select the proper units within this tool.*



Tool Settings

Tab	Tool Settings	Effect
General	Units	Some files do not contain unit information. This setting will allow you to specify the units of those files. *US Survey Feet, International Feet, Meters*
	Thin Incoming Data	Enable to load partial amounts of each data file based on the 'Nth Point' value.
	Nth Point	Value for Thinning Incoming Data. *Value of 2 means 50% of data will load, 3->33%, 4->25% and so on.... 10->10%*
	Thin Static Scans by Distance	Thins out the over-saturated region around the system (proximity to the scanner origin) while maintaining points farther away. This will generally reduce the point count for a static scan by 80-90%! *<low, medium, strong> for STATIC scans with an ORIGIN ONLY.*

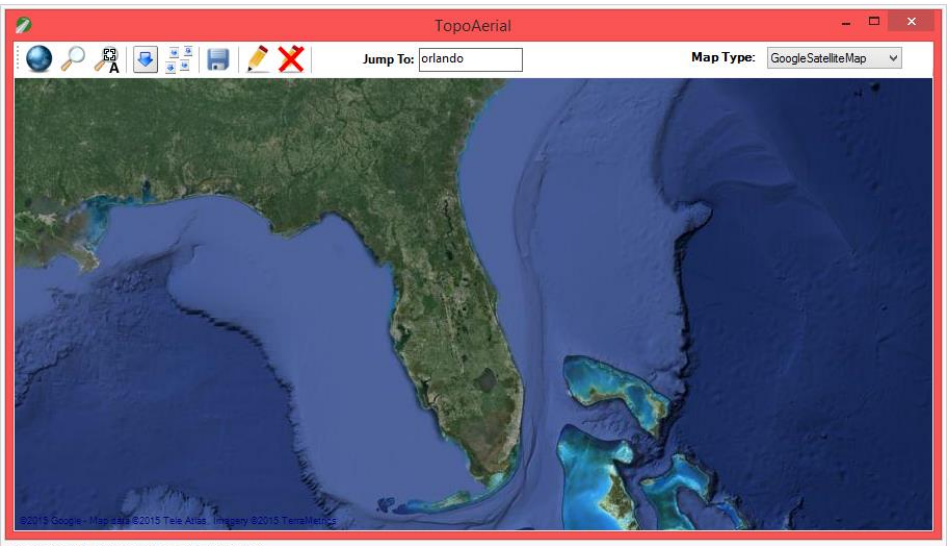
	Data Noise Thickness	Some tools will use this noise tolerance value for calculating the ground elevations. Rule of Thumb: take a thin cross section (0.5'-1') on a hard surface and measure the depth of noise you see (min to max) and then use this value for your 'Data Noise Thickness'. For example, if a thin slice of my road surface is 0.25' thick, then this value will be used because this is my ideal flat surface value. Also, adjust for grassy areas as well because they will be different thicknesses in most cases.
	Limit Incoming Points	Limits the total point loading capacity to the specified value. This is to cap the memory used by loading in data so that the TopoDOT tools can still function. The value entered will vary slightly between workstations.
LAS Files	RGB Range	LAS files can contain RGB information within their internal format. Select the correct range so the RGB information will be read correctly within TopoDOT.
	RGB Export	There are 3 options: RGB, Plane Deviation, Color Contour. *See Export Points on how to export points.* • If RGB is selected then current RGB values will be used during export. • If Plane Deviation is selected then the current color scheme (defined by ACS, min & max values) will be used to override RGB values during export. *See Plane Deviation Color Scheme to change color scheme and see Display Settings to calibrate color-gradient to point elevations in 'Distance From Plane' tab.* • If Color Contour is selected then RGB values will be overridden with the current 'Colored Contour' settings. *See 'Colored Contour within Display Settings.*
	Classify by Flight Line	When enabled, normal LAS file classifications will be replaced by the flight line information. *The point cloud file must have internal flight line information for this to take effect.*
RIEGL Files	Pulse Deviation Limit	When enabled, this may reduce LiDAR artifact points base on their internal deviation values.
DOT Files	Load As One File	When loading DOT files, if enabled, the data will appear as one single file in the point cloud manager. If disabled, each individual file that was previously used in creating the DOT will appear in the point cloud manager.
	Optimize Exported DOT Files for Visualization	This will allow optimization during the manual export process. *See Export Points on how to export points. If you are looking to Optimize a set of DOT files please see Optimize Files For Visualization*

Load Aerials

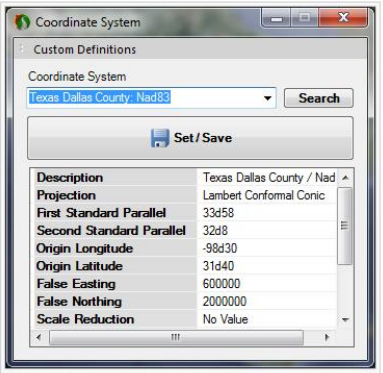
Purpose

Allows user to reference files from the public database of aerial images such as GoogleSat and BingSat. These aerials are imported into MicroStation™ and provide a rough estimate of project environment and terrain. Aerial Images are most helpful when starting out projects with no CAD work or extraction, because the images can be used as templates for setting up an extraction process. *Also, a great alternative to not having High Resolution Calibrated Aerial Images!*

- *There are a few options to choose when using TopoAerial images within MicroStation. Please see How To's for details:*
    1. *Temporarily transfer what is currently being displayed within the field of view (FAST)*
    2. *Temporarily transfer detailed images of what is currently being fenced in (SPEED depends on size of region being fenced)*
    3. *Permanently save a fenced in region as *.jpg images for rastering, see HOW TO's below for more details. You can either use the Pencil-Fence or the Red Fence (hold both alt and right click).*



TopoAerial window once the tool is launched



Window for selecting Coordinate System

Tool Settings

Load Aerials

Tool Settings	Effect
Select Coordinate System	Select Project State Plane or UTM Zone
Zoom & Auto-Zoom	Zoom to Project area once coordinate system is chosen or Auto Zoom.
Transfer	Import displayed (current zoom and area) Aerial Image into Microstation™ view. *This temporarily pulls the imagery into the design environment.*

 Transfer Detailed Tiles	Tile and Import (zoomed-in-images) the displayed Aerial Image into MicroStation. This will also work in relation to the 'Fence Project Area' feature. *This temporarily pulls the imagery into the design environment.*
 Save	Save a specific Aerial Image as .jpg with a sister file .jpgw (world file)
 Fence Project Area	Creates a fence around a region of interest. 'Transfer Detailed Tiles' will use this fence to bring in necessary zoomed-images.
 Remove Project Area	Deletes the created fence *if one exists*.
Jump to	Type City, State, Zip to Manually move to an area of interest
Map Type	Select source and type of Image/Map, such as 'GoogleSatelliteMap'

Note: Other details are described below in How To's

How To's

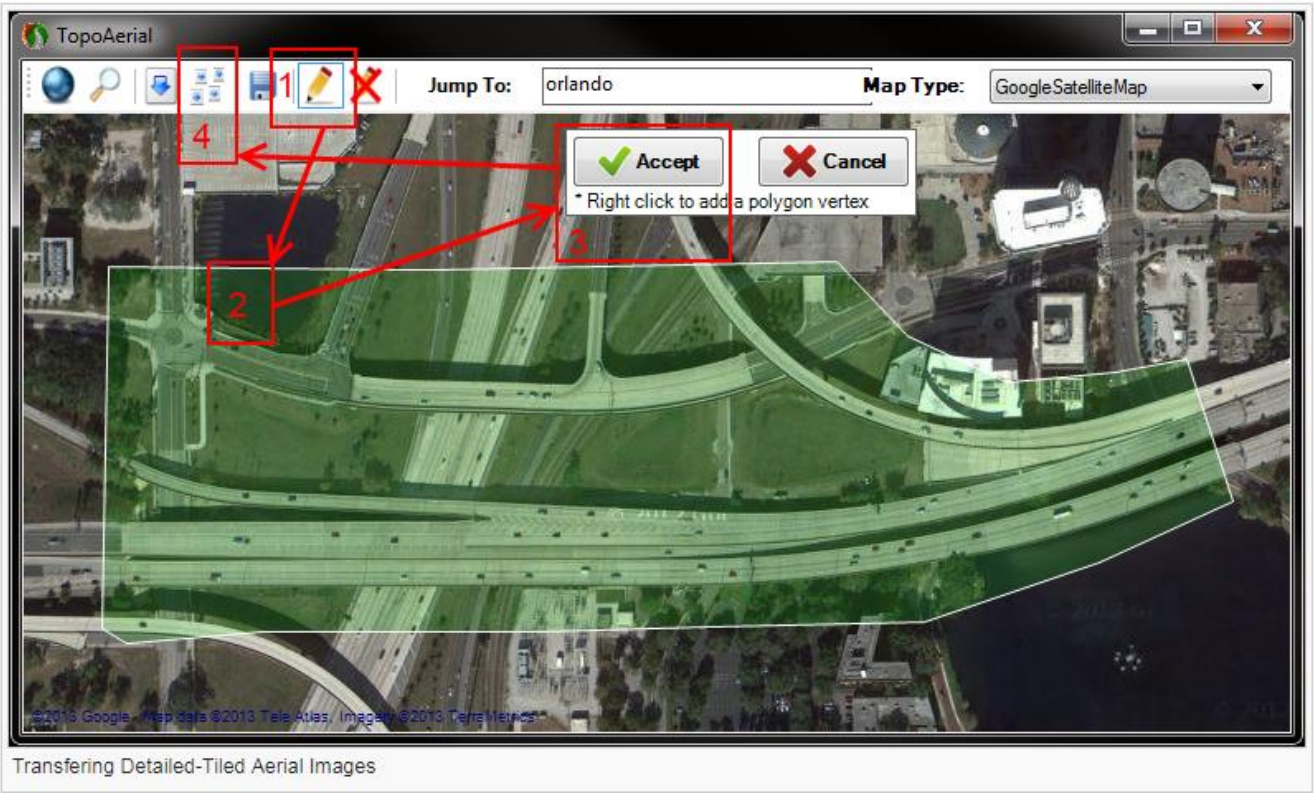
To Transfer Aerial into Micro Station view

1. Activate Tool by clicking Icon. *A 'TopoAerial' window will appear.*
2. Click 'Select Coordinate System' *(option window appears)* and chose a system corresponding to project. 'Search' State Plane or UTM Zone.
3. Click 'Set/Save'
4. Now, click 'Zoom' to zoom into project area or simply scroll the mouse wheel to zoom in.
5. Isolate a Aerial region within the TopoAerial window, and click 'Transfer'.

Note: Make sure there is CAD elements or LiDAR Data within the design file, such as 'Mapped Scan Icons' or a .las file, in-order for TopoAerial to locate project area (Image elevation will be placed at a set distance below these elements/data to avoid covering project). Also, the project should be geo-referenced to use these photos effectively. Image is flat, so use while in TopView only.

To Transfer Detailed-Tiled Aerial Images into Micro Station view

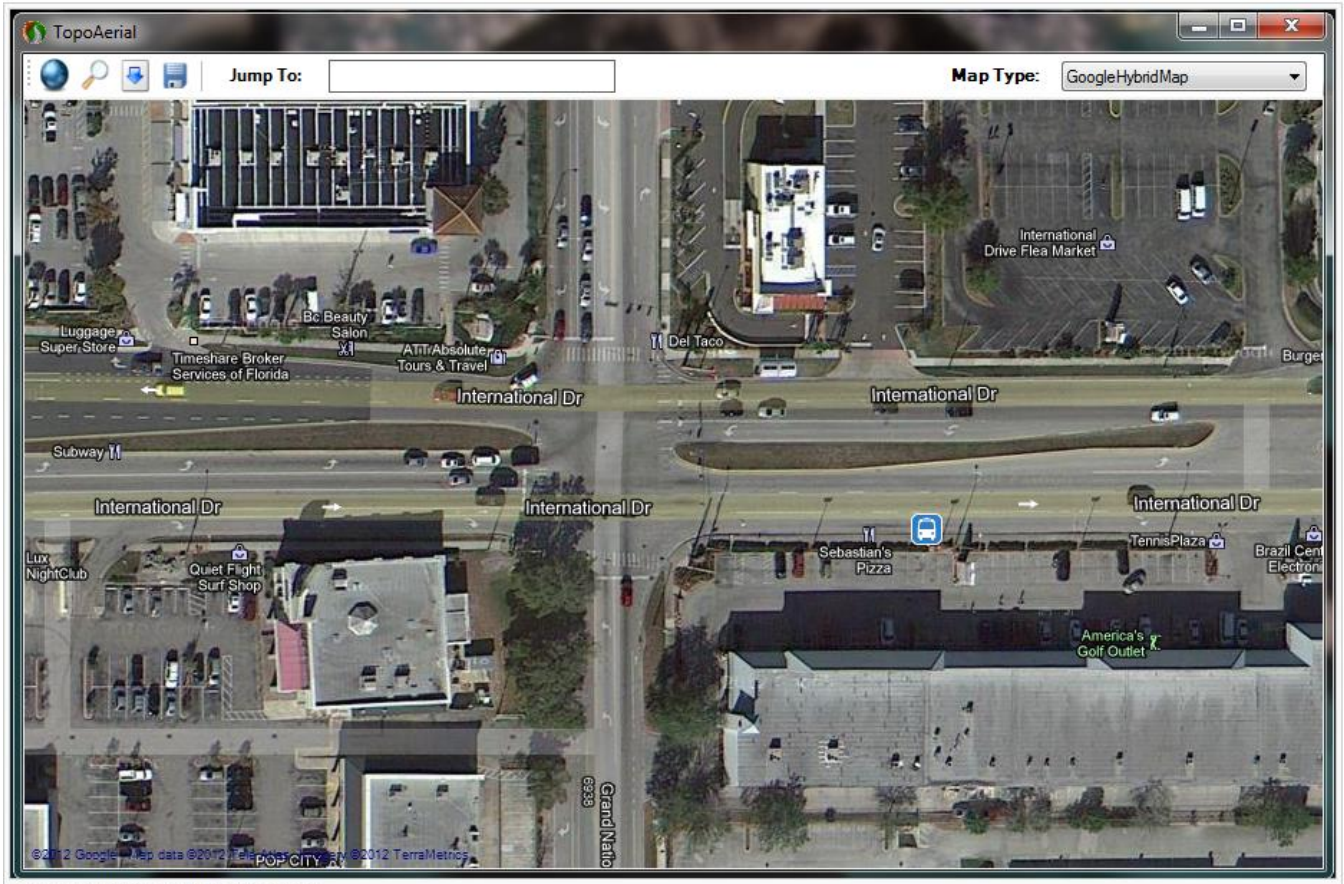
-
    1. Activate by clicking the icon. *This will prompt an 'Accept and Cancel' sub window*.
    2. Now draw a fence around the region of interest. *Right-Click to add polygon vertex*.
    3. Once the fence is or desired results or completed, simply click 'Accept'.
    4. You are now ready to proceed with importing detailed images (which are tiled) from this region. Select the 'Transfer Detailed Tiles' feature to import into MicroStation.
        - Detailed-Tiled Images are going to be loaded within the MicroStation view and will be indicated via a progress bar.



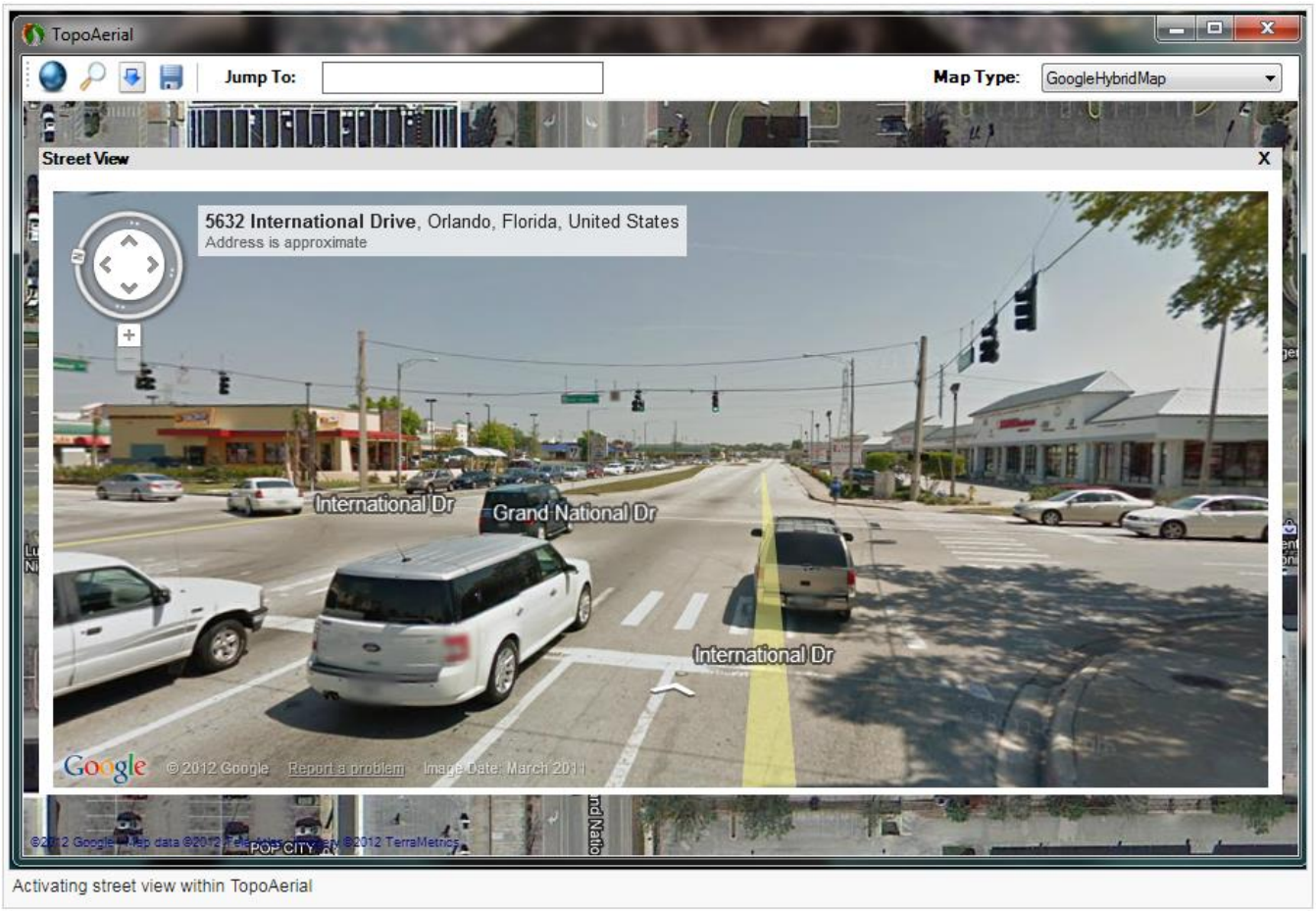
Transferring Detailed-Tiled Aerial Images

To Utilize Street View

- 1. Select the Aerial region of interest.
2. Hold 'Ctrl' and 'right-click' on the selected Aerial region to obtain a Street view.



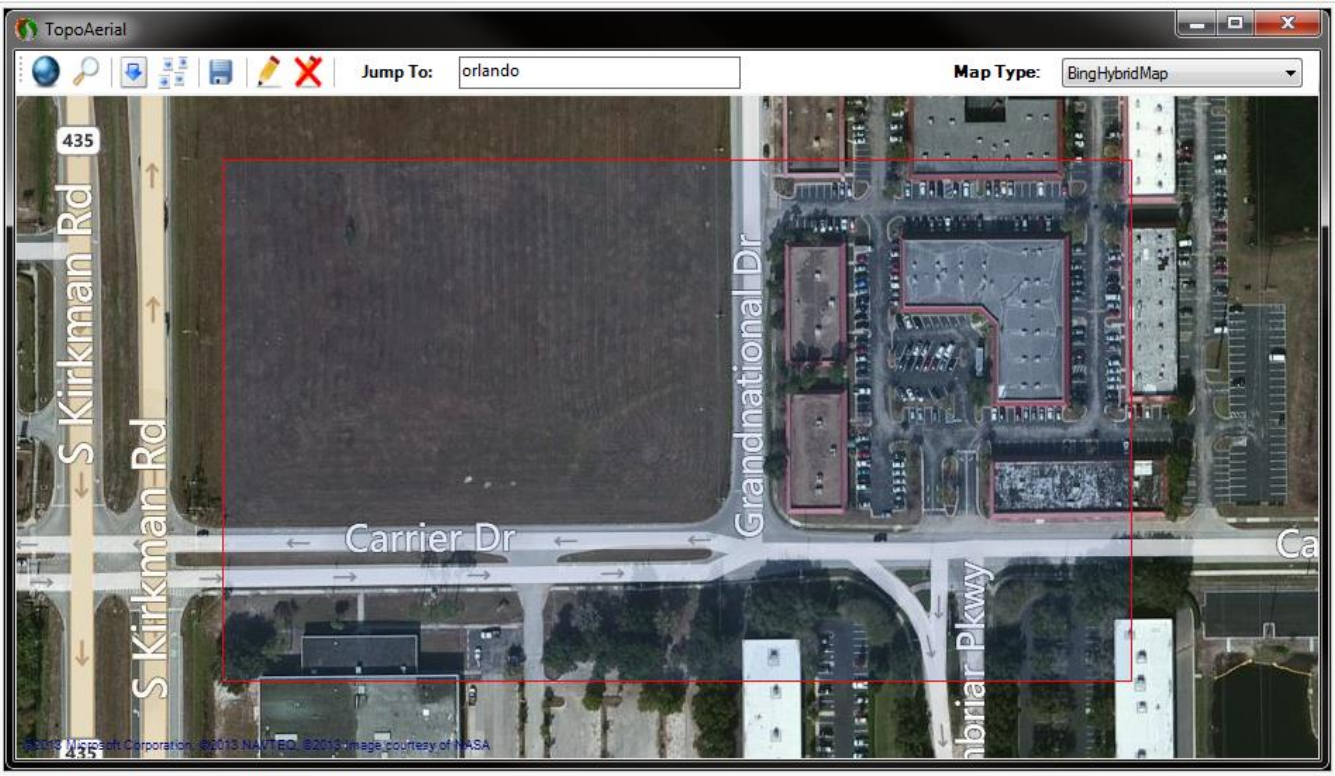
Activating street view within TopoAerial



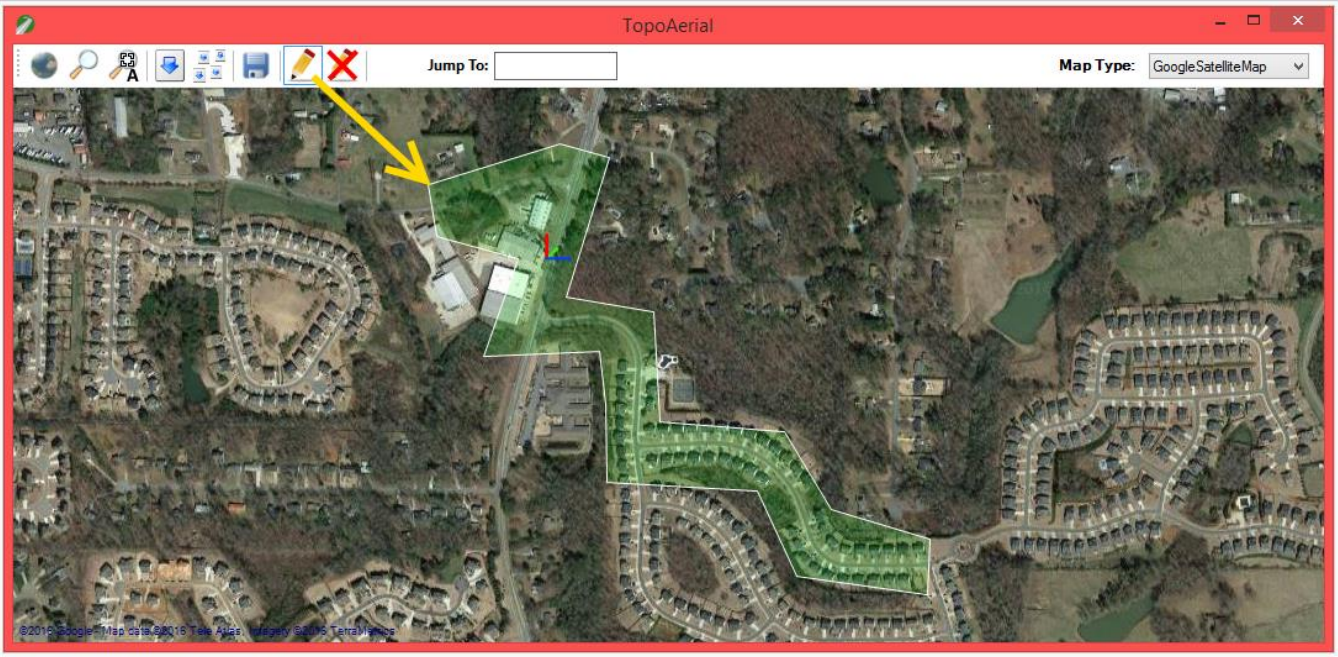
Activating street view within TopoAerial

To Save Aerial Image for future Rastering into Micro Station

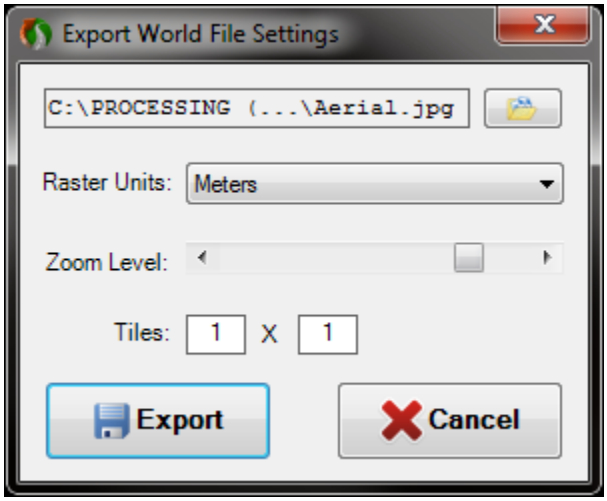
- 1. First, select or box-in region to be saved. Hold 'Alt' and hold 'right-click', then drag to box in region of interest. *Red box will appear.*
    - An alternative method for selecting an area is to use a 'Fence' (pencil looking icon). *This method will ignore the specified tile-array within the tool and save based on a preset determined tile size.*
2. Now, click 'Save' Icon, to save this Image as a .jpg file. *A corresponding sister-file will auto save as a .jpgw file.*
3. Click, Save. *Export World File Settings - Window will appear*.
4. Choose 'Save Location'
5. Choose 'World File Units'
6. Choose 'Zoom Level'. *How detailed the images will appear.*
7. Choose 'Tile Grid' (ONLY applies to the Alt-Rclick method and does not apply to the pencil-fence method). *How many images there will be total. So, a 1 X 1 = 1 Image, 2 X 2 = 4 Images, 3 X 2 = 6 Images.*
8. Aerial Image(s) have been saved as a .jpg with .jpgw sister-file(s).



Window for Selecting Region (red box) to 'Save'



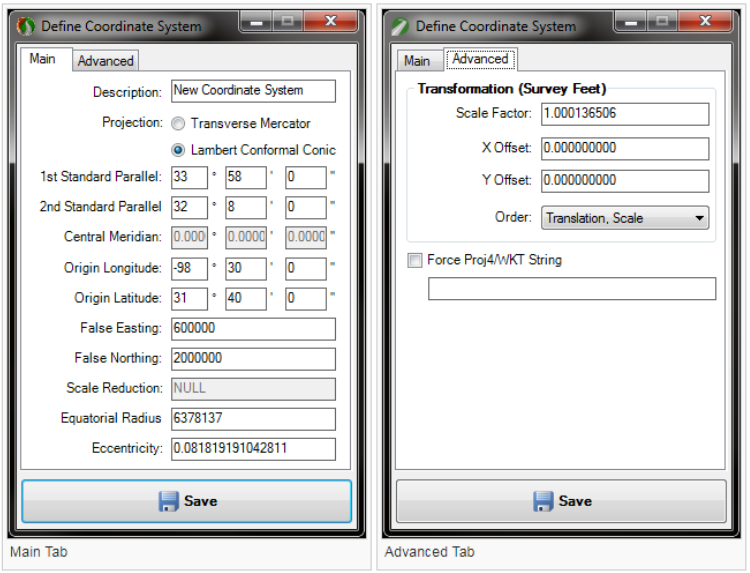
Alternative method for selecting an area using pencil-fence

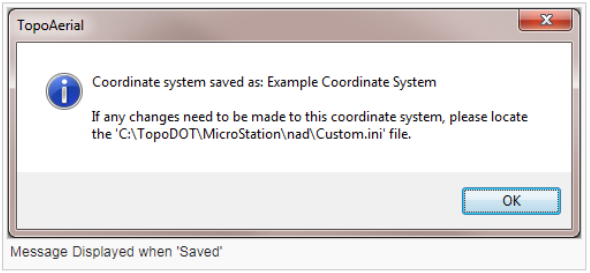


Window for choosing directory, units, zoom and amount of tiles

To Create a Custom Coordinate System

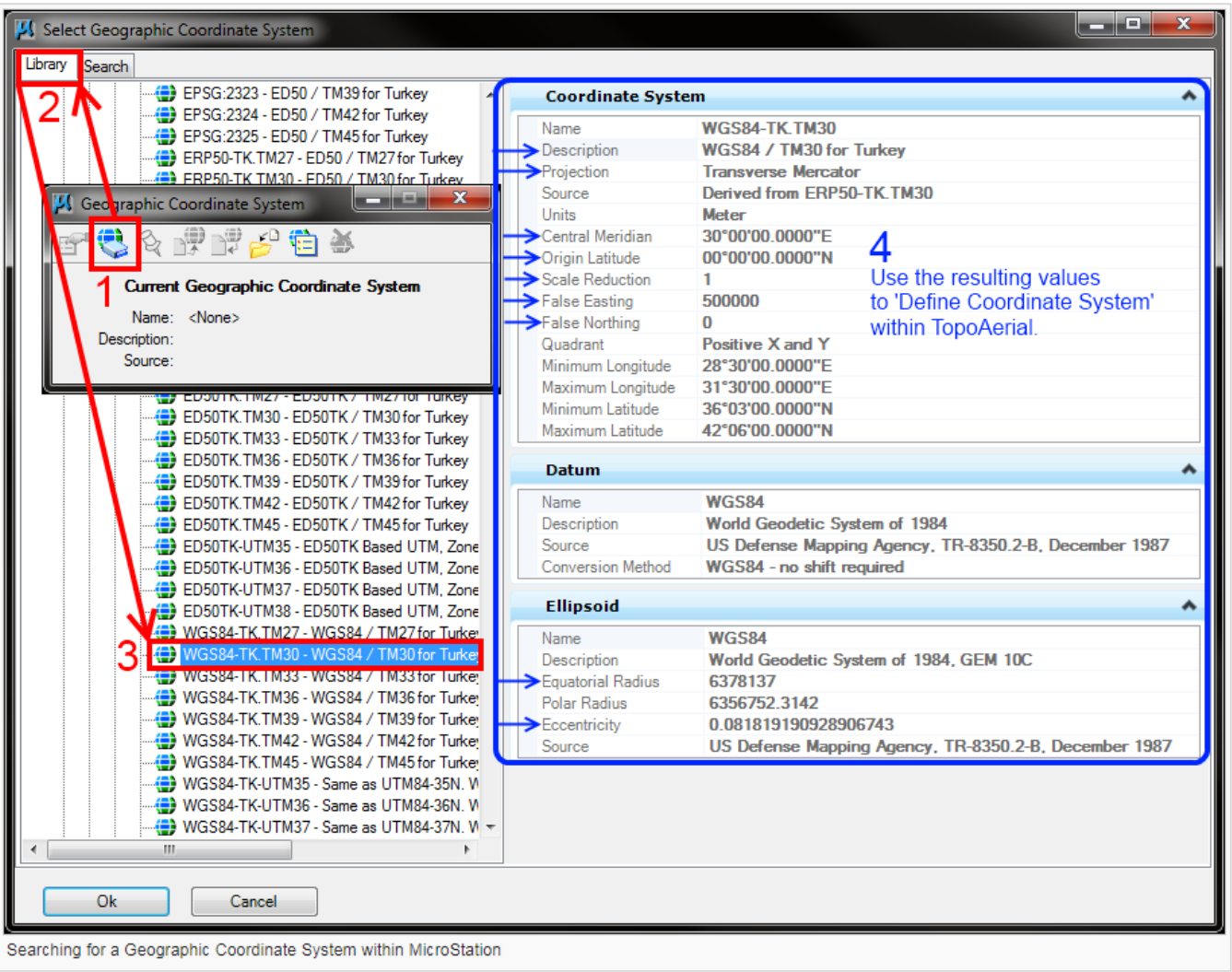
1. Activate Tool by clicking Icon. *A 'Topo Aerial' window will appear.*
2. Click 'Select Coordinate System' and then click on the 'Custom Definitions.'
3. Navigate through the Main and Advanced tabs and fill out all that apply.
4. Once done, click 'Save' which will store the defined coordinate system within an existing file.
   - *The file location C:\TopoDOT\MicroStation\nad\Custom.ini*
5. Now, go back to 'Select Coordinate System' and chose the newly created coordinate system.





Message Displayed when 'Saved'

Note: If you don't have the coordinate system or ellipsoid values then try locating it within MicroStation's library: Go To 'Tools->Geographic->Select Geographic Coordinate System.' Now click the 'From Library' Icon and find the corresponding coordinate system information either by searching or selecting from the Library Tree.



Searching for a Geographic Coordinate System within MicroStation

To Raster a Saved Aerial Image into Micro Station

1. You must have a saved Aerial Image and World File in-order to accomplish this. See above 'How To' for this.
2. Locate image(s) using 'Raster Manager'.
   - Please Search 'Attach Raster' using MicroStation's Help Content. *Help/Contents/Search/raster attach. Then search for saved Image file.*

To Remove Aerials from working view

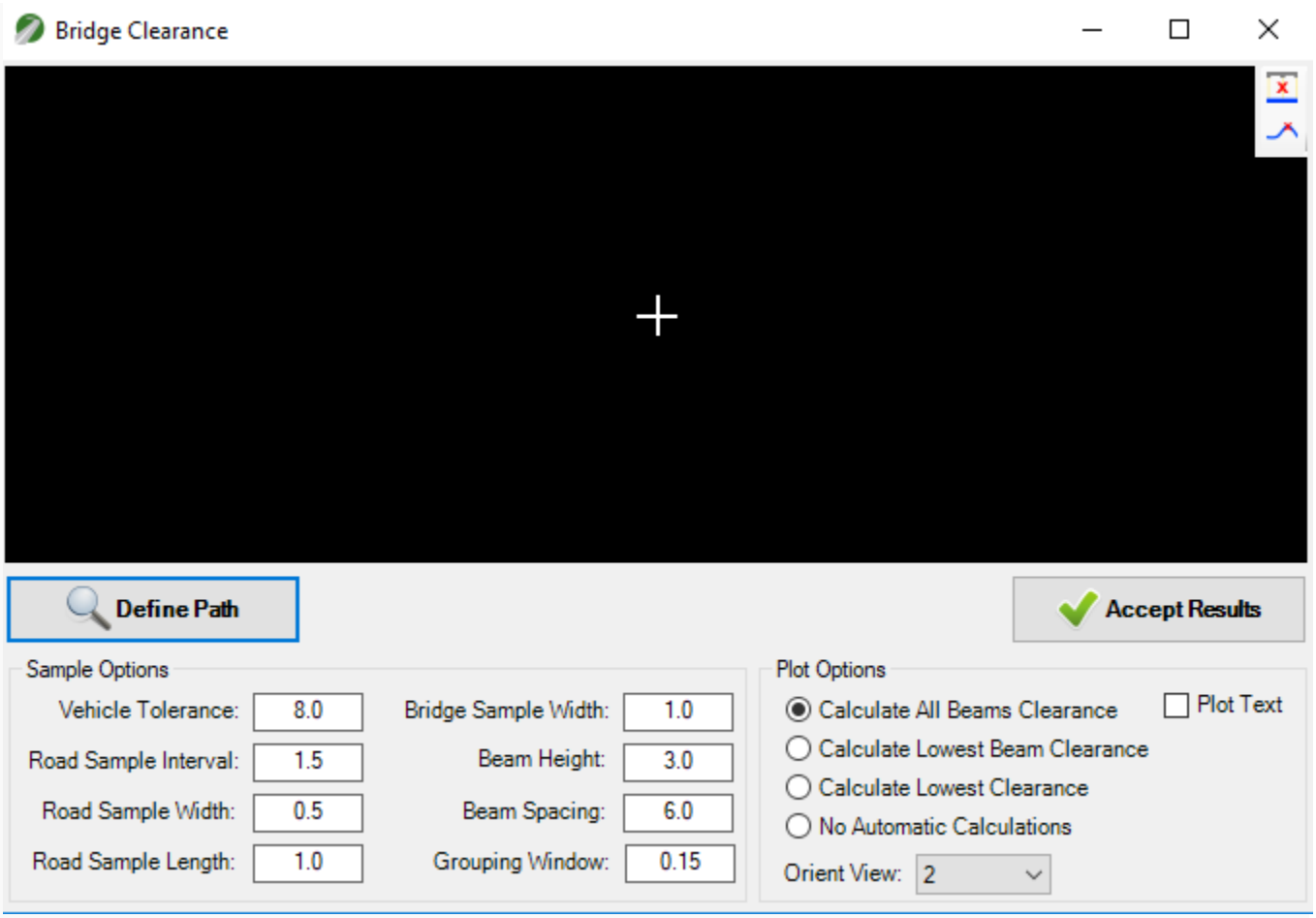
1. See Remove Aerials for removing temporarily transferred aerials, not rastered aerials.

Bridge Clearance Tool

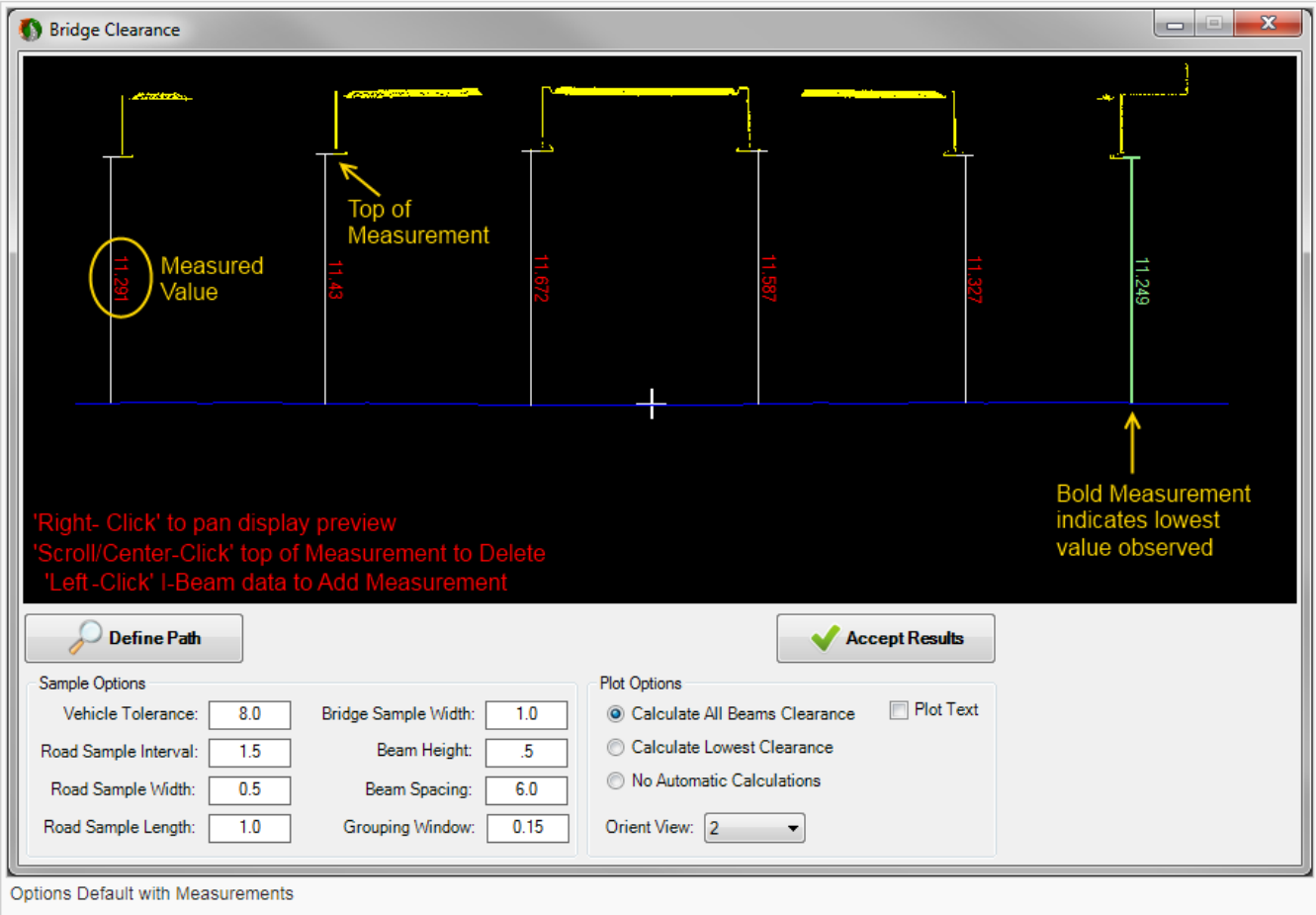
Purpose

Used to measure Bridge Clearances from features such as I-Beams or other features (*Powerline clearance to road for example, within the How To's below*).

Note: If working in the CONNECT environment and dimensional elements are appearing properly please note that your dimensional styles need to be modified from default if not already set up. Step 1) Go to Dimension Styles and create a style then name (example: 'dimensions'). Step 2) under the Text tab make sure 'Font, Height, Width' are set appropriately. Might have to disable 'Font'.



Options Default with No Measurements Yet



Options Default with Measurements

Tool Settings

Sample Options

•	Tool Settings	Effect
Sample Options	Vehicle Tolerance	Enter the height of any vehicles that may be included with the sampling process. *If this is set too high, as to interfere with actual bridge data, then there will be no data to display within the preview.*
	Road Sample Interval	Interval of sampling for the ground elevation. The intervals will take place along the direction defined.
	Road Sample Width	Width of data to sample for ground elevation (Perpendicular to path direction)
	Road Sample Length	Length of data to sample for ground elevation (Parallel to path direction)
	Bridge Sample Width	Enter the width of the Flange of interest
	Beam Height	Enter the height of the beams of interest
	Beam Spacing	Enter the spacing of the beams of interest

	Grouping Window	Boundary(width) dimensions for sampling data, which calculates the low and high points of bridge profile
	Define Path	Activates the placement of path for sampling process to take place
	Accept Results	Accepts all measured values within the preview window
Plot Options	Calculate All Beam Clearances	Finds all available beams and places measured values.
	Calculate Lowest Clearance	Finds only the lowest clearance beam and places a measured value.
	No Automatic Calculations	Places no measured values. User will place dimensional values manually.
	Orient View	View used for the orientation of the dimension elements (related to 'Plot Text'). *This value will most likely be at the bottom of the dimensional element.*
	Plot Text	Enables auxiliary measured values to be placed at the base of each dimensional element. This Text will be oriented according to the 'Orient View' setting.
	 Clear Clearance Measurements	Removes all clearance measurements
	 Remove Road vertex	Removes selected road vertex

How To's

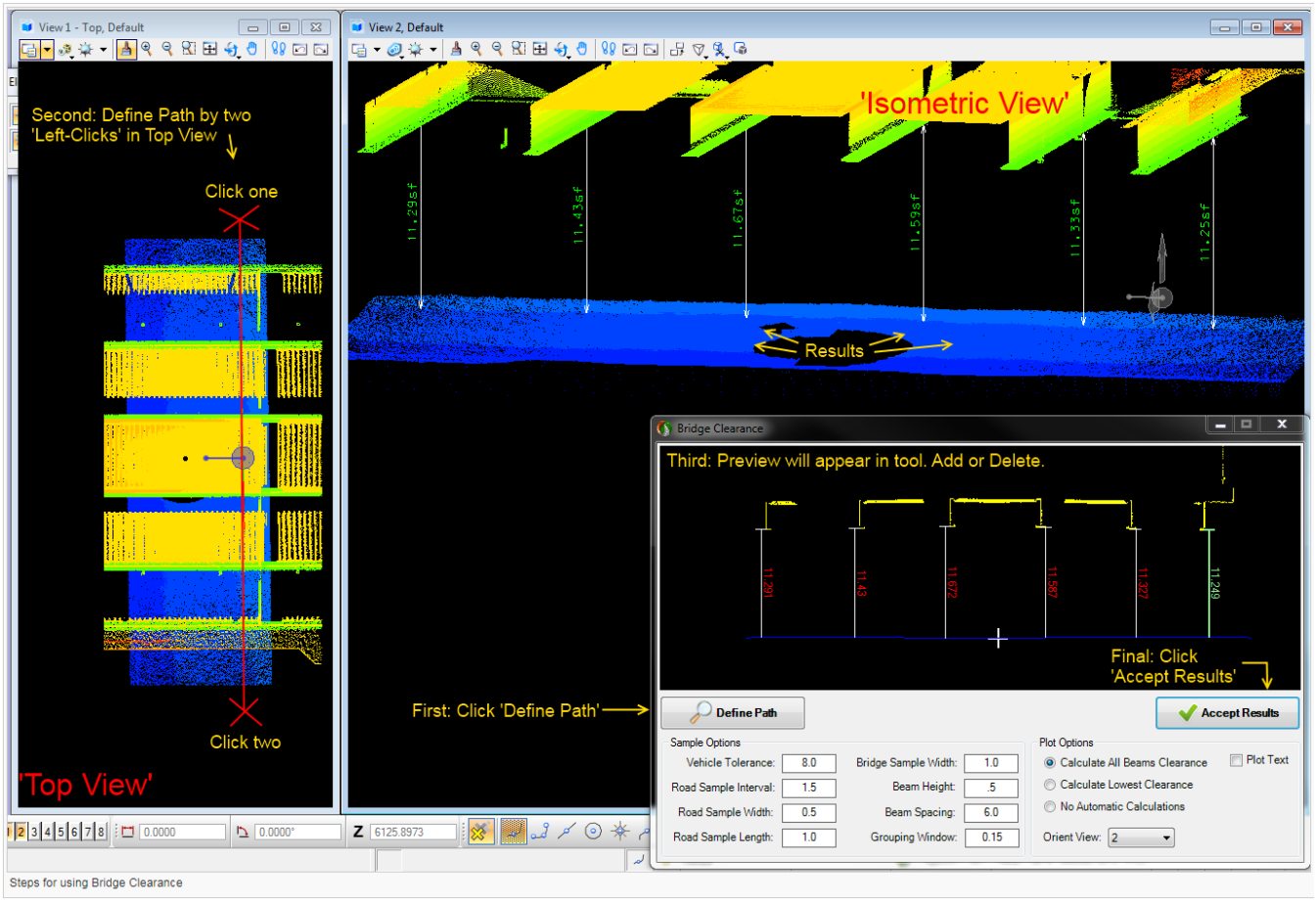
To Measure Bridge Clearances

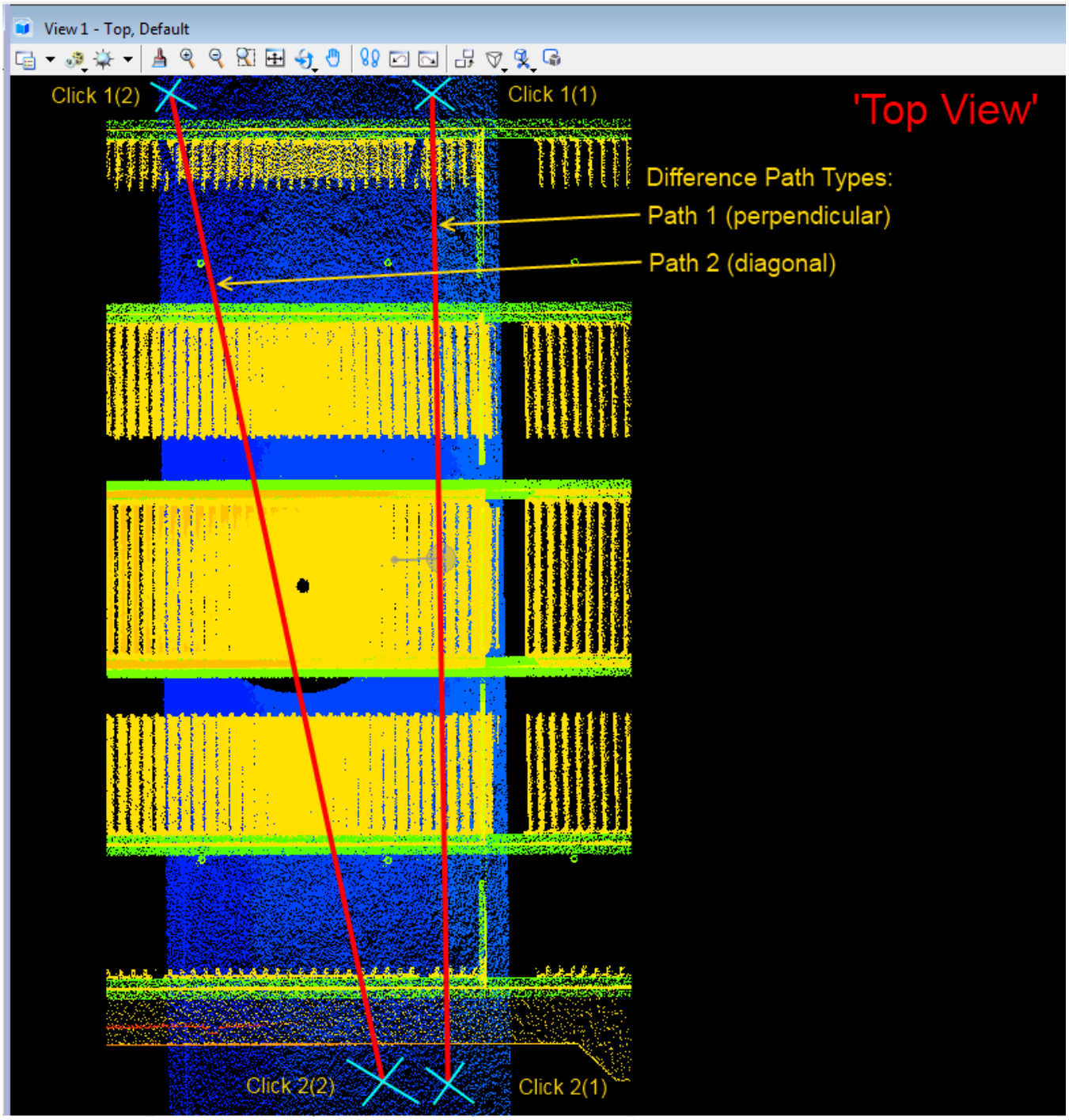
1. Isolate or locate beam data and have multiple perspective views ready for reference. *Mainly a top view perspective and cross section Cross-section by Points*
2. Activate tool by clicking icon, *window will appear*:
   - Proceed to entering the specific settings within the 'Sample Options'
     1. Use MicroStation's measuring tool to quickly measure profiled features of the bridge.
     2. This step may not be necessary, but it will produce the best results for unique bridge profiles.
   - Select which results to be plotted in 'Plot Options'
     1. See Tool Settings for details
3. Now click, 'Define Path'.
   - Within top view, define the path across (perpendicular) the bridge. To define, click once on one side of the bridge and then click a second time on the other side of the bridge. *Make sure to include the entire width of bridge, because these two clicks define the boundary of sampling (which is basically a cross section)*
   - This path does not have to be perpendicular. For instance, if a road passes diagonally underneath the bridge, then a path parallel to the road may be defined. But, the larger the deviation from perpendicular, the more tilt there will be in the bridge profile; therefore, 'No

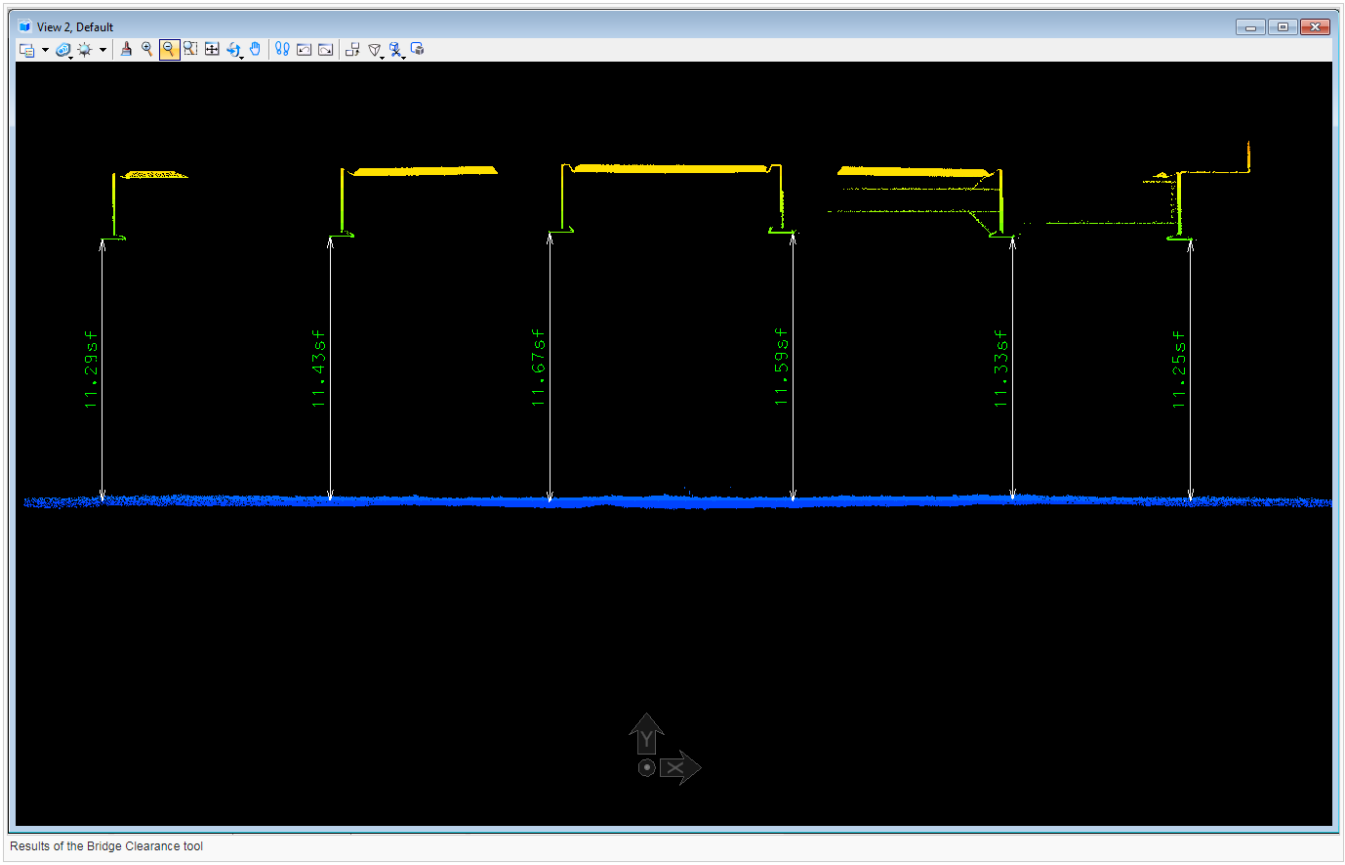
Automatic Calculations' may have to be chosen for the 'Plot Options'. This way, user can manually input measurements based on what he/she sees in the preview window.

4. Bridge profile will appear within the tool preview window. '*Right-Click' to pan, 'Scroll/Center-Click' top of measurement to delete, 'Left-Click' to add measurements.*
   - The Lowest value measured will be bold and highlighted a 'mint-green' color. *Reference only, and is visible only in preview*
   - Examine analysis and make sure the results are satisfactory. Add (for features missed or for non-beam features such as pipe) or Delete (for duplicates or unwanted features) as necessary.
5. Once QA/QC has been carried out, click 'Accept Results'.
6. The dimensional elements and measured values in the preview have now been placed within the design file.
7. To repeat, define new path and follow steps.

Note: When Results are accepted, the lowest calculated value will automatically be placed onto your 'clipboard' so you can proceed to pasting this value within another document: .csv, .txt, .doc, etc.




View 1 - Top, Default
Click 1(2)
Click 1(1)
'Top View'
Difference Path Types:
Path 1 (perpendicular)
Path 2 (diagonal)
Click 2(2)
Click 2(1)

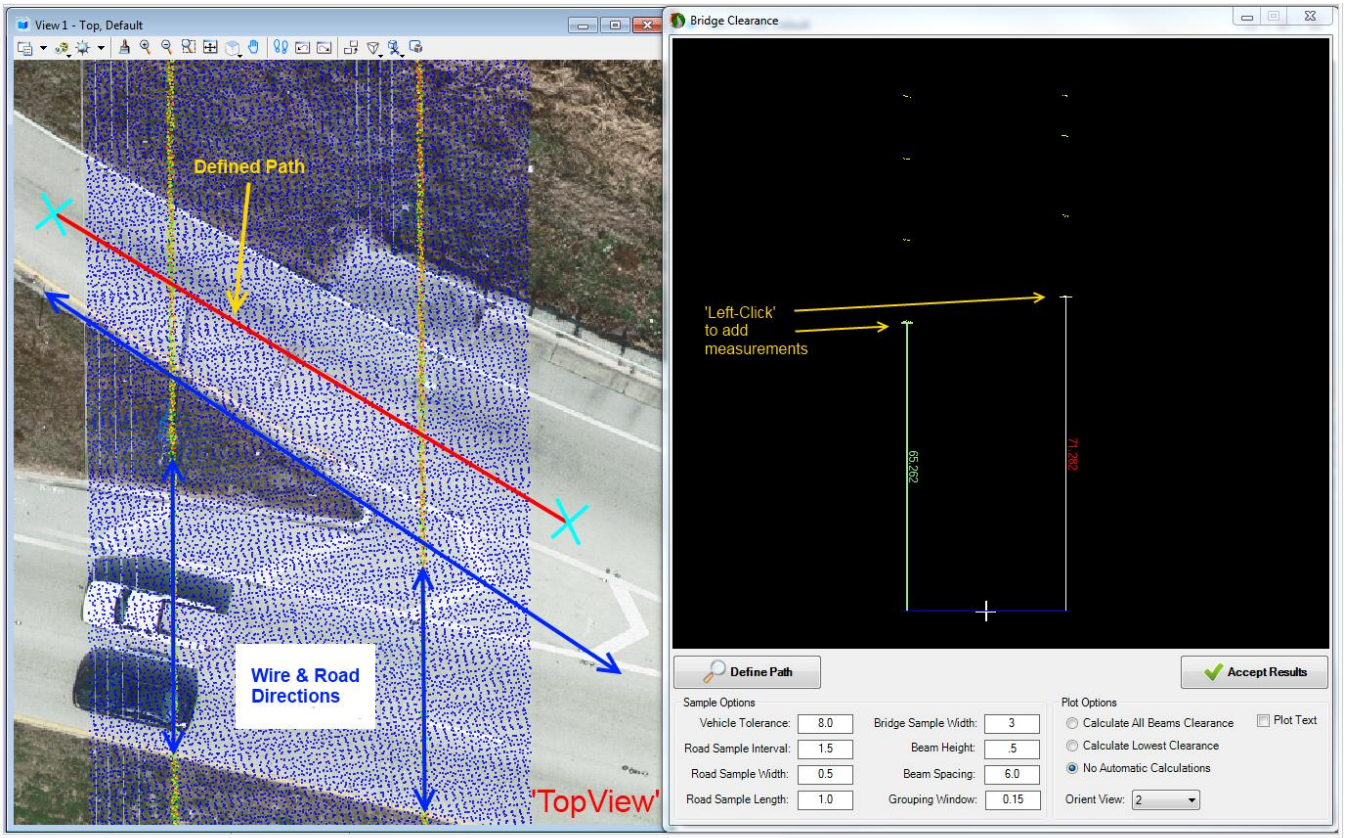


Results of the Bridge Clearance tool

Note: This tool is not limited to Beams on Bridges, so please feel free to apply this tool to other applications such as Power Lines, Overhead Signs, Cranes, Pipes on Bridges, etc.

To measure Wire to Road Clearances

1. Follow steps as above, but instead choose 'No Automatic Calculations' for 'Plot Options'
   - Now add measurements to preview (which will be cross section or profile of wires)
2. Click 'Accept Results' to place measurements



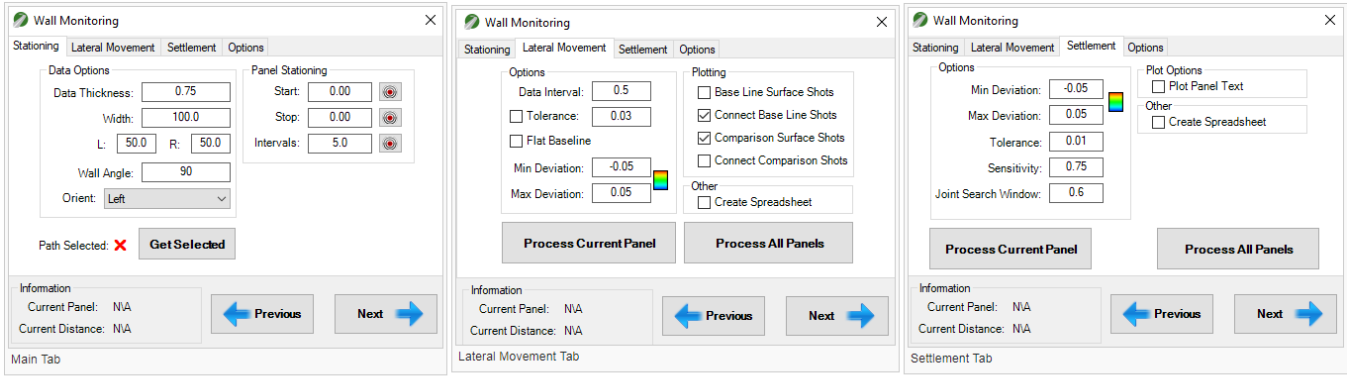
Applying Bridge Clearance tool to overhead power lines over a diagonal section of road

Wall Monitor

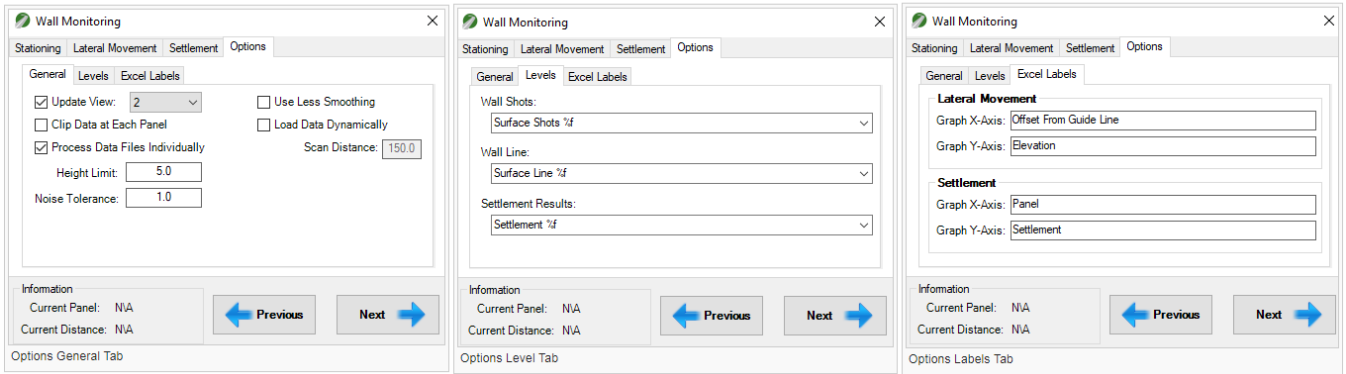
Purpose

Allows the user to monitor multiple wall surface data sets for deviation and settlement. There are two ways to use this tool: one is to use a guideline that the user traces out on an ordinary flat wall (this method is limited to lateral movement only) and two is to use a CSV Stationing file on a paneled-retaining wall (this method will allow for lateral and settlement calculations).

For a detailed explanation of what this tool is capable of on a paneled retaining wall please see our tech-note here.



OPTIONS TAB



Tool Settings

All Tabs- At bottom of tool

Tool Settings	Effect
Information	Displays the Current Panel/Position/Station and the Current Distance. *Note: Tool starts everything from the beginning of the line unless specified otherwise.*
Previous Panel	Move down along the guide line. *This tool is for navigation purposes only and will not process.*
Next Panel	Move up along the guide line. *This tool is for navigation purposes only and will not process.*

Data Tab

Tool Settings	Effect	
Data Options	Data Thickness	Enter the thickness (in working units) of the desired slice to take at each step/panel/position for sampling purposes. The thickness depends on the density of the scan, surface conditions, flatness. What would constitute a higher valued sample? -> sparse scan data, deeply ruffled/grooved/textured surface, relatively flat surface when sampling. What would constitute a lower valued sample? -> very dense data, smooth surface texture with no grooving depth, relative surface is not flat and more organic in nature (rock face for example).
	Width	Search parameters on how far to search for data from the guideline. This will look (perpendicularly from guide line) left and right a specified distance. *This will override values within the L/R text boxes.*
	L: / R:	Modify left and right distance from the guideline to process. *For example, this allows you to look 5 units left and 10 units right.*
	Wall Angle	Reference angle to use for processing wall information. If the wall has a 45-degree tilt (throughout) from vertical then match this, if the wall varies +/-20 vertical then keep reference angle at 90 degrees.
	Orient	Orientation along guideline. This should be pointing towards the surface from the scanned side.
Path Stationing	Clip Data	Turns off all point cloud data except for current station. This is essentially going to clip the data as tool moves up and down guideline.
	Start	A user defined position along the path on where to start. This will apply when 'Auto' extraction is used.
	Step	Station interval along guideline (in working units). Tool will move down the guideline at this interval.
	Position	A user defined or 'Current' position/location along guideline.
	Path Selected	Select path for tool to follow and click 'Get Selected' to Activate. *Note: compatible elements include a poly lines, arcs, circles, blocks/shapes and bsplines.*

Lateral

Tool Settings	Effect	
Options	Data Interval	Enter spacing in which points are placed in the profile/cross-section (working units). These will be the points going up the wall.
	Tolerance	This relates to the deviations defined. If enabled, measured deviations within this tolerance will be colored white.

	Flat Baseline	Measurements will be from a plumb/vertical baseline (the guideline will be your 'zero' and a vertical baseline will originate from here at each panel/position). *This feature should be used if you want to compare a single wall file to a vertical plumb line. Disable this feature when comparing two files of the same wall.*
	Min / Max Deviation	Minimum/Maximum Deviation used for coloring shots. Everything outside of the min/max deviation will be assigned the min/max color.
Plotting	Base Line Surface Shots	Check to enable shots to be plotted on the base line data.
	Connect Base Line Shots	Check to enable profile line to be plotted on the base line data.
	Comparison Surface Shots	Check to enable shots to be plotted on the comparison data.
	Connect Comparison Shots	Check to enable profile line to be plotted on the comparison data.
Other	Create Spreadsheet	Create excel spreadsheet for the entirety of the wall and each individual cross section. There will be a main sheet that displays all deviation points to the user as if he/she were looking at the wall (wall is essentially flattened out so curving walls will now seem flat). The following sheets will show each cross section with graphical and indexed deviation values.
	Process Current Panel	Processes only the current position.
	Process All Panels	Processes the entire length of the selected guide line at the specified stepping intervals.

Settlement

Tool Settings	Effect	
Options	Min / Max Deviation	Deviation used for coloring any resulting arrows and deviation text. Everything outside of the min/max deviation will be assigned the min/max color.
	Tolerance	This relates to the deviations defined. If enabled, measured deviations within this tolerance will not be identified but you will still get a flat deviation element (green).
	Sensitivity	Sensitivity level for locating panel joints. Higher values will make the tool less sensitive to noise. Default value is 0.75 (design file in feet).
	Joint Search Window	This distance value defines how far to search for other comparison joints, if 2 joints are found and each are from a different file than they'll be coupled for comparison. The value here should be larger than the observed movement but smaller than the span in between repeat joints (on the same wall/data). *NOTE: For a paneled wall the distance in between the horizontal*

		joints are usually around 5 feet while the comparison movement is only 0.02-0.06 feet.
Plot Options	Plot Panel Text	This will assign panel text to each position/station. Text will be 'View Independent'.
Other	Create Spreadsheet	Create excel spreadsheet for the entirety of the wall and each individual cross section. There will be a main sheet that displays all deviation points to the user as if he/she were looking at the wall (wall is essentially flattened out so curving walls will now seem flat). The following sheets will show each cross section with graphical and indexed deviation values.
	Process Current Panel	Process the current position only. *Make sure the data is displayed.*
	Process All Panels	Process the entire length of the selected guide line at the specified stepping intervals. *Make sure the data is displayed.*

Levels Tab

	Tool Settings	Effect
Levels	Wall Shots	Designate level for this feature extracted. *The file name of the corresponding point cloud will also be included within the level name. For example, if the point cloud file name was 'PT123' and the designated level was 'Surface Shots' then the resulting level produced will be 'Surface Shots PT123'.*
	Wall Line	Designate level for this feature extracted. *File name applies just similarly, see above.*

Options

Tool Settings	Effect	
General	Update View	View used to display the current cross section. This will update as we move along the guideline.
	Clip Data at Each Panel	Clip or Turn-off point cloud data except for the current station/position/panel.
	Process Data Files Individually	Process each data set separately.
	Height Limit	Max height based off the reference wall, 'plane', to consider points as noise and ignore. For example, if there is vegetation next to the wall and the guideline is within 1' of the wall data, then we can specify a value of 2' for a height limit to help filter out noise. Remember that this value is normal to the to the wall angle you specify.

	Noise Tolerance	Relative to the amount of noise in the data. Higher tolerance number corresponds to higher level of noise. *This should be the depth of the 'fuzz/noise' normal to the wall sample. To measure this, take a thin (usually the size of your 'data thickness' or sample) cross section and measure.*
	Use Less Smoothing	Less Smoothing of the result but higher chance to pick up noise. If the wall varies such as a rock face would then 'enable' this, but if the wall is relatively flat and consistent then 'disable'.
	Load Data Dynamically	Loads necessary data, samples, and then Unloads that data at every cross section.
	Scan Distance	Determines the relevance of scan data based on the specified distance from the scan position to the cross section.
Levels	Wall Shots	level for Wall Shots
	Wall Line	level for wall line
	Settlement Results	level for Settlement Results
Excel Labels	**Lateral Movement**	The string specified here will be used for labeling purposes
	Settlement	The string specified here will be used for labeling purposes

Pre – How To's

There are two types of movement that can be monitored within this tool: Lateral and Settlement. Lateral movement is within a horizontal reference frame. Settlement is a vertical shift in a wall. Please see our tech-note which further explains this notion, tech-note
Note: A stationing file (CSV format) is NO LONGER supported to replace a physical CAD guideline. The 'Path Selected' (guideline) tells the tool where the wall(s) or panel(s) is in horizontal space and the 'Wall Angle & Orient' tells how it is tilted and oriented to the guideline.

Lateral Movement Cases:

1. Movement relative to a Flat Baseline (one file or one wall)
   - Here we will be comparing one vertical wall to a flat baseline so that the reference point for monitoring will always be against a vertical surface.
2. Movement relative to a Flat Baseline (two or more files/walls)
   - Here we will be comparing two or more vertical walls to a flat baseline so that reference points for monitoring will always be against a vertical surface. Typically, you'd have 'Process Data Files Individually' ENABLED.
3. Movement relative to another Point Cloud (two or more files/walls)
   - Here we will be comparing two vertical walls relative to themselves. Typically, you'd have 'Process Data Files Individually' ENABLED.

Settlement Movement Case:

1. Settlement relative to another Point Cloud (two or more files/walls)

- Here we will be comparing two walls relative to themselves so that the shift can be tracked.

How To's

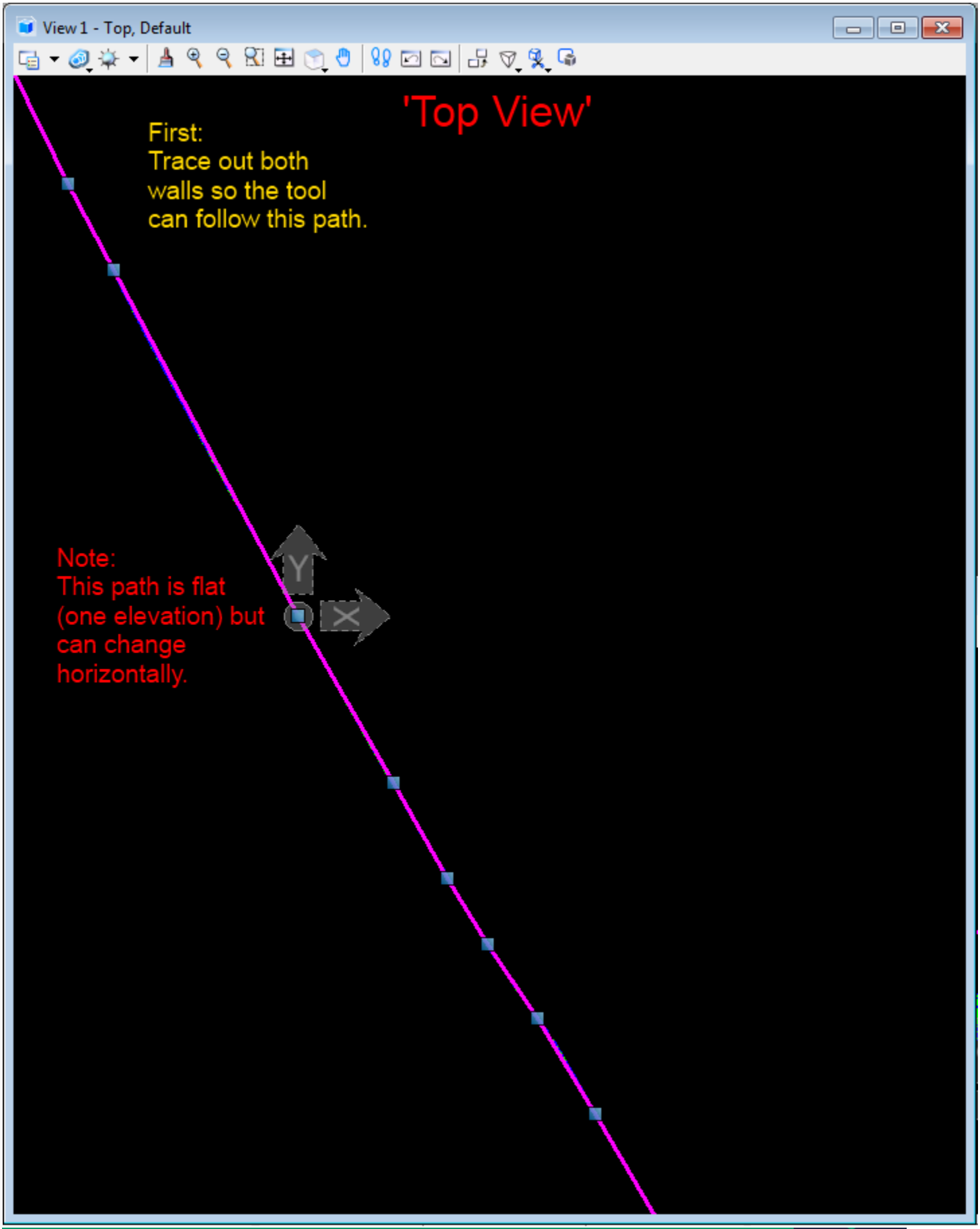
Method 1:

Extracting a Grid of Colorized Spots on an Ordinary Wall Surface (Gradient based on deviation)

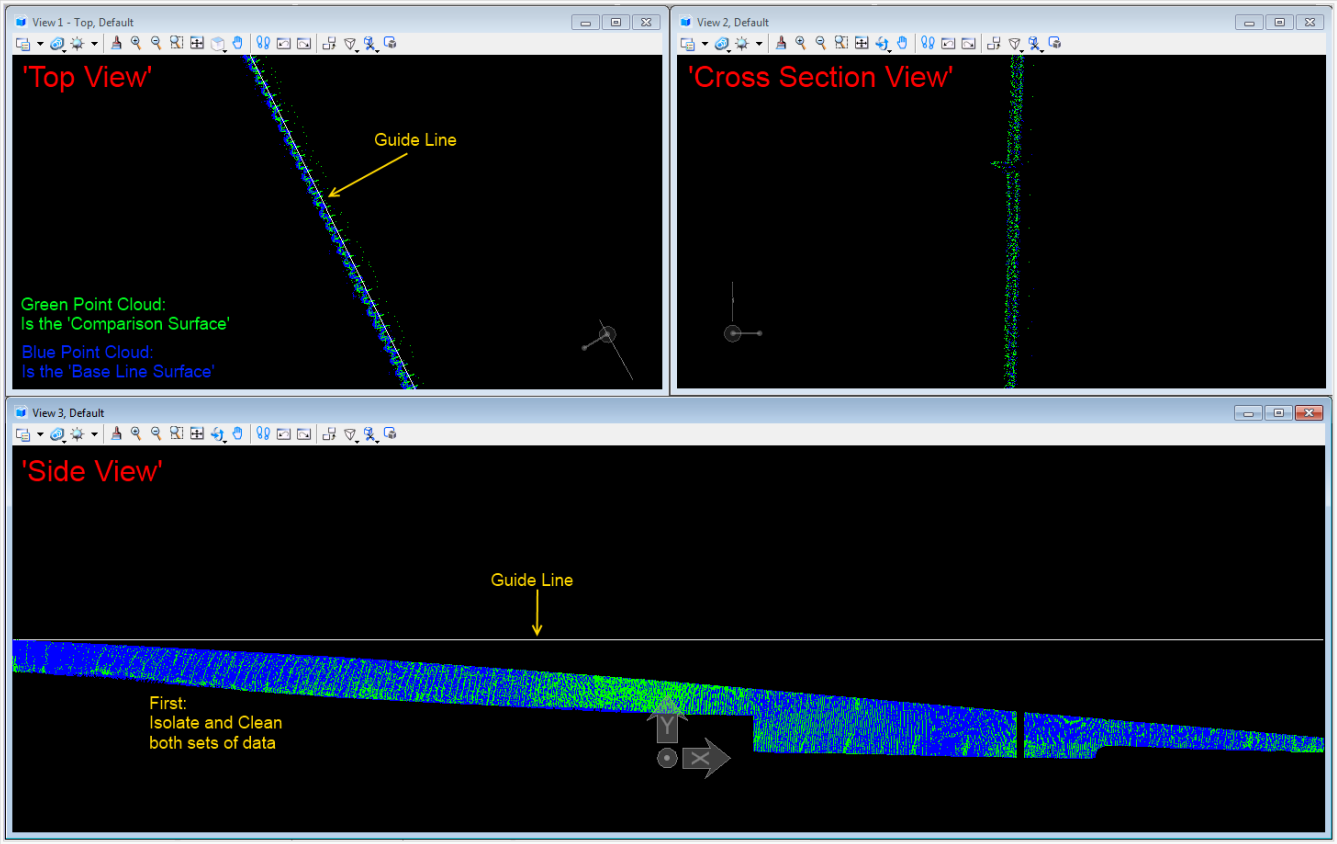
1. Load the first set of data, which will be the 'Base Surface'. Then load the second set of data, which will be the 'Comparison Surface'. *See Point Cloud Manager for file order and color*
   - Now isolate a region of the surface to process. Go ahead and place a fence around the section of wall and crop the data outside the fence for further isolation. See Crop Points Outside Fence.
   - The preferred mode of display would be 'Individual Scans'. See Display Settings for details.
   - Also, it is possible to crop each data set individually and then export as a .las or .dot point cloud file. These newly created files can now be loaded and processed. *Order of loading still matters, see above.*
2. Now, place a guide-line down the center of the surface data (this should be done in Top-View and at an elevation close to the data). The guide-line should be flat, and only change horizontally to follow surface. *This will ensure a tight fitted guide-line for the tool to follow*
   - Also, a fence can be fitted to this guide-line to help crop out data beyond the wall surface. See Fence Along Element and Crop Points Outside Fence. *This would take the place of fencing and cropping earlier, but make sure the guide-line is also a good center line*
3. Open the Wall Monitor tool by clicking the icon. *Tool interface will appear*
   - Select or Highlight Guide-Line and then Click, Move Up or Down to activate the positioning of the ACS. *This will be like the Road Extraction tool.*
   - Enable 'Comparison Surface Shots'. *Recommended because this will be the surface that has deviated from the base surface*
   - Make sure to also Enable 'Process Data Files Individually' within the Advanced Tab.
   - Disable 'Clip Data, Base Line Surface Shots, Connect Base Line Shots, Connect Comparison Shots'. *Optional*
   - If necessary, adjust Data Options: 'Width, Wall Angle, Data Interval, Step, Etc.'. *Optional*
     - *The Wall Angle will appear as a magenta-colored line and should be lined up with the surface data within the cross section view or slice. 90degree value will be straight down or at 6 o'clock.*
4. With the guide-line highlighted, click 'Process Entire Guide Line'. This will automatically run along the path and process the entire surface data.
5. To inspect, click 'Move Up' or 'Move Down' and examine each profile within the 'Update View' (usually view 2).
6. Once done, the tool will have placed a grid-like pattern of surface shots or Point Elements.
   - These Point Elements will be colorized based on the deviation value. Red (Max Deviation), Green (Neutral Deviation), Blue (Min Deviation). This way, there will be a color gradient to visualize and spot any areas that have deviated the most.

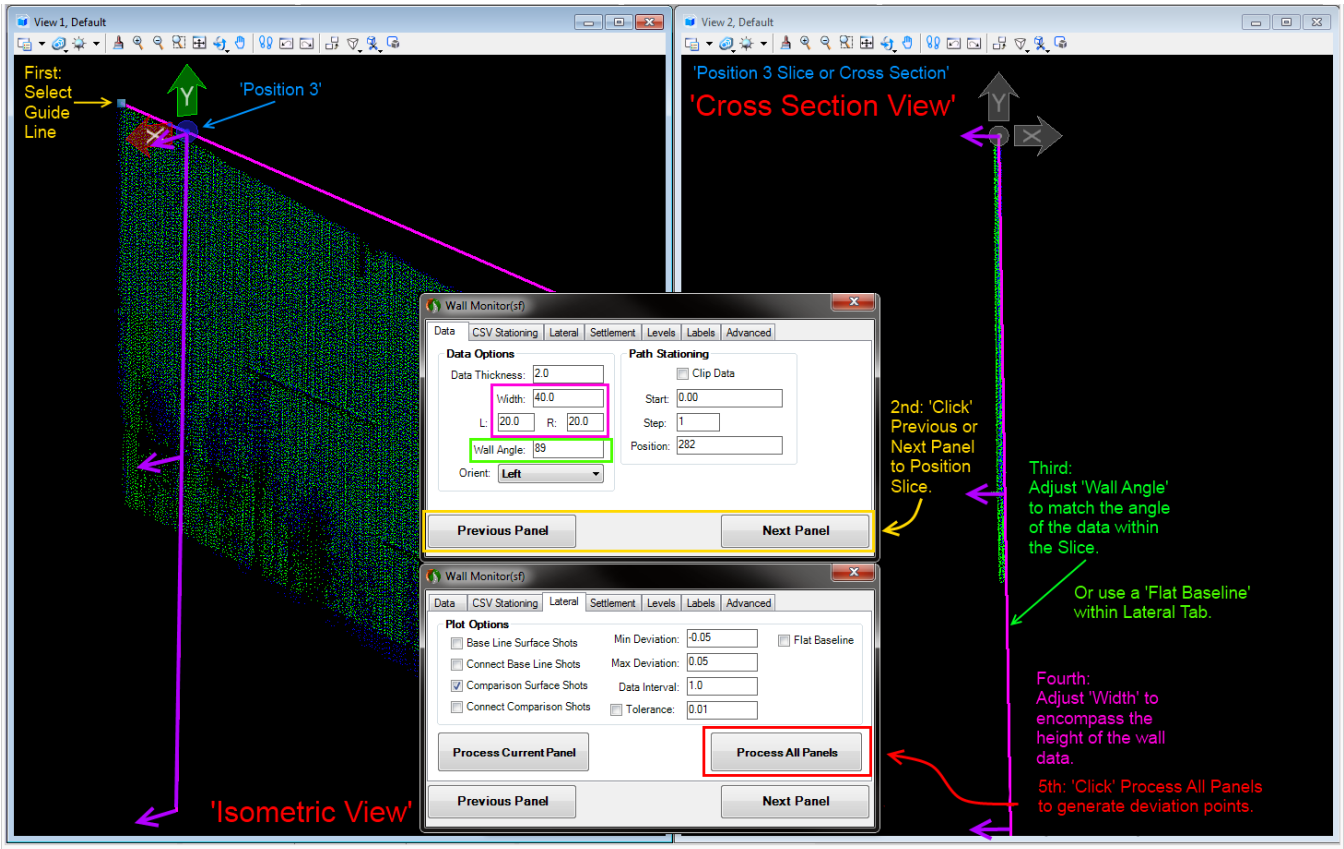
Note: If the Surface Shots or Point Elements are appearing as White, either the point falls outside of the deviation values or the point is no good (possible data noise could pull a point away from the surface to create a false deviation value).
Note: Make sure that when loading data, the order of this data is known, because the first point cloud file will

be the 'base surface' while all other files proceeding, will be the 'comparison surface(s)'. See Point Cloud Manager to view order and color of data files. Technically, three surfaces can be loaded, with one being the base and two being the comparison surfaces.
Note: If you decide to combine different days of a cleaned wall within a DOT format, then make sure to Disable 'Load As One File' within the DOT Files tab of the File Settings feature.

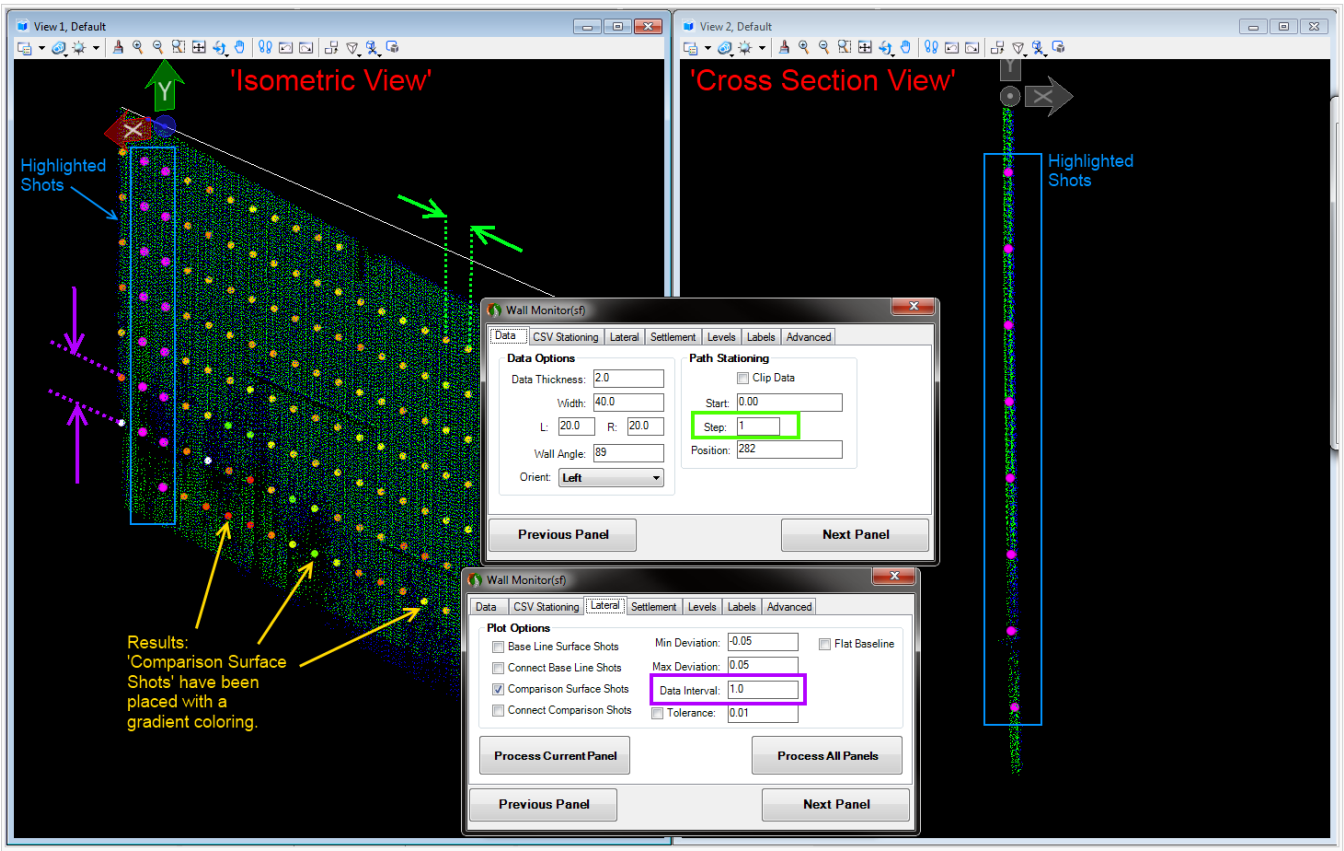

View 1 - Top, Default
'Top View'
First:
Trace out both
walls so the tool
can follow this path.
Note:
This path is flat
(one elevation) but
can change
horizontally.
Y
X

Creating a Guideline in the Direction of the surfaces

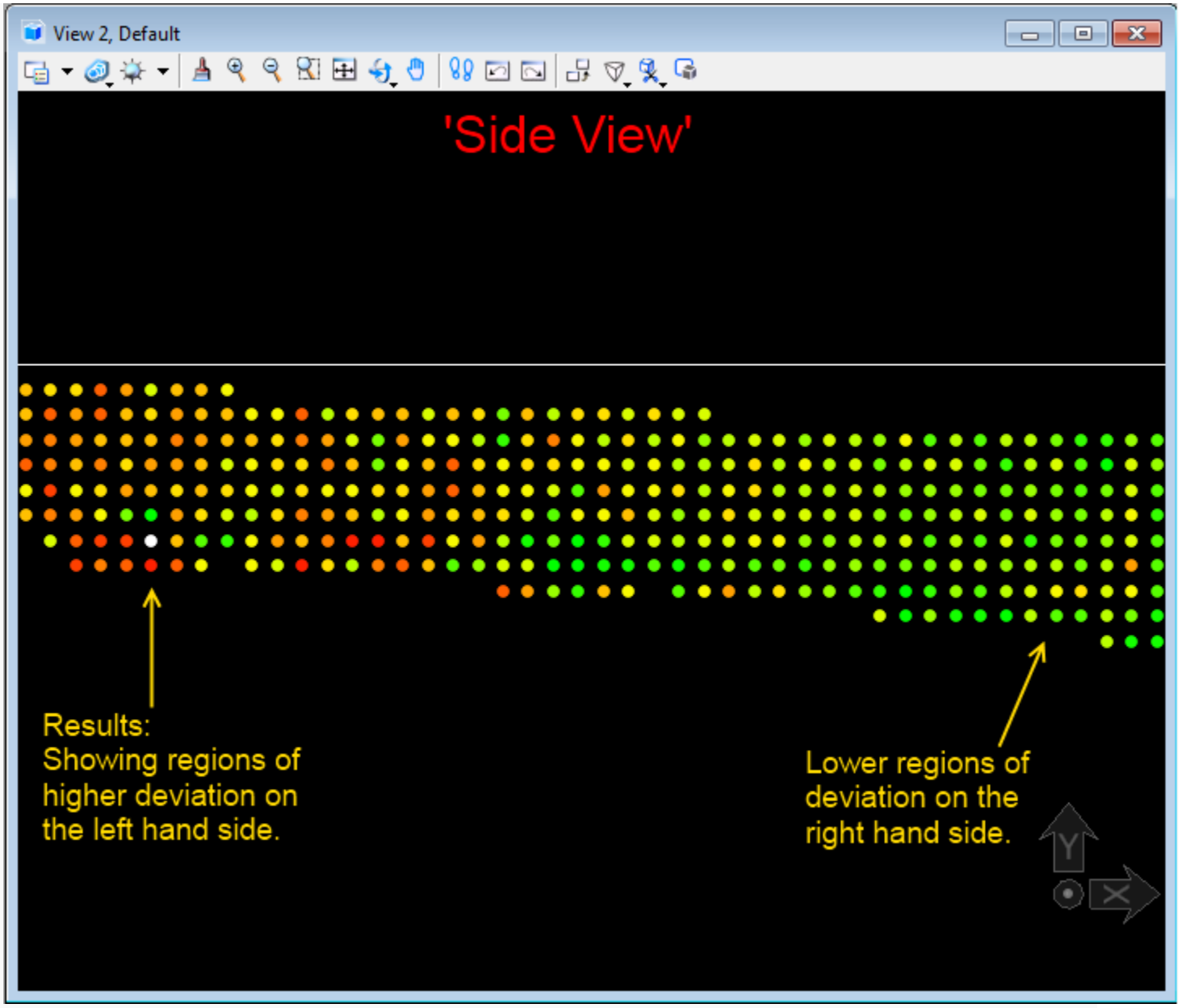




Using the tool interface



Resulting Surface Shots



Results: Showing areas of high (red) and low (green) deviation

Method 2:

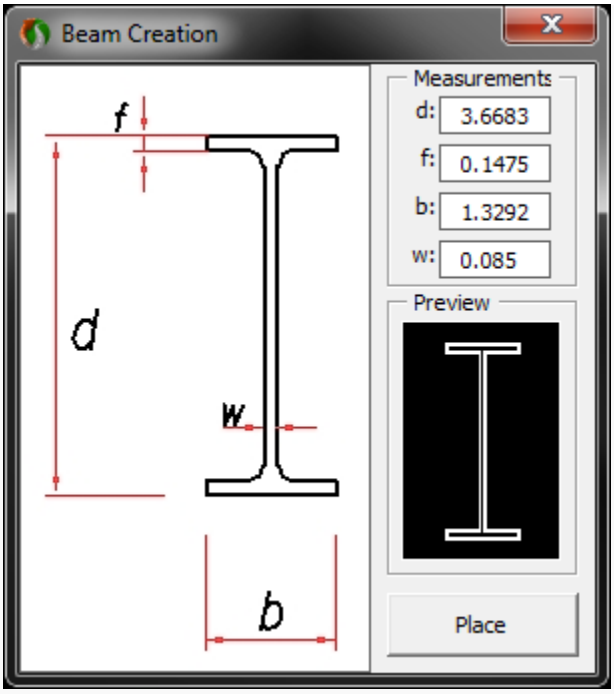
Extracting Lateral and Settlement movement using CSV Stationing file for Panel-Retaining Walls

1. Contact office for setting up this tool. (407) 248-0160

Beams

Purpose

Used to create beams based on specified dimensions that have been measured.



Window of tool Options

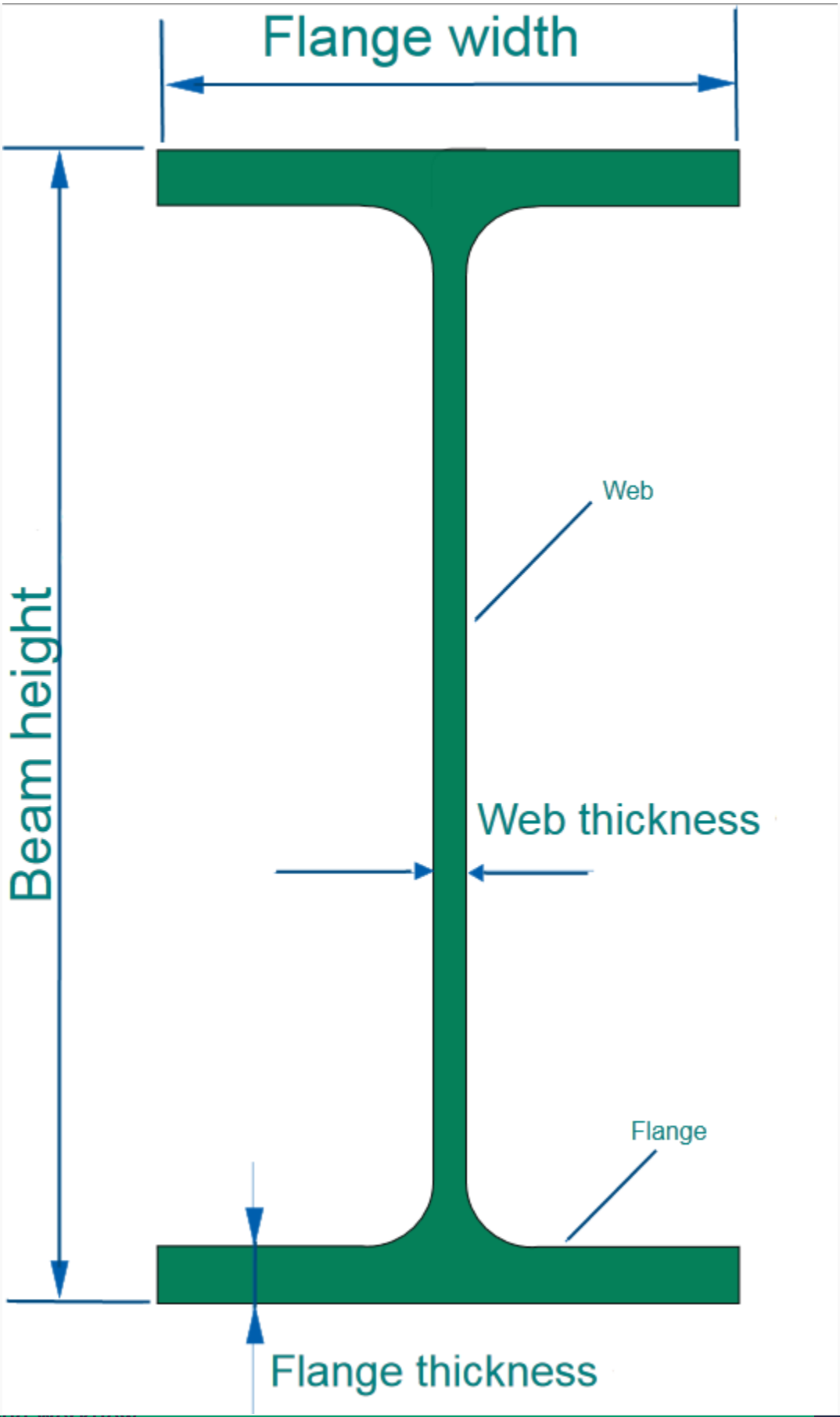

Flange width
Beam height
Web
Web thickness
Flange
Flange thickness

Image of dimensions

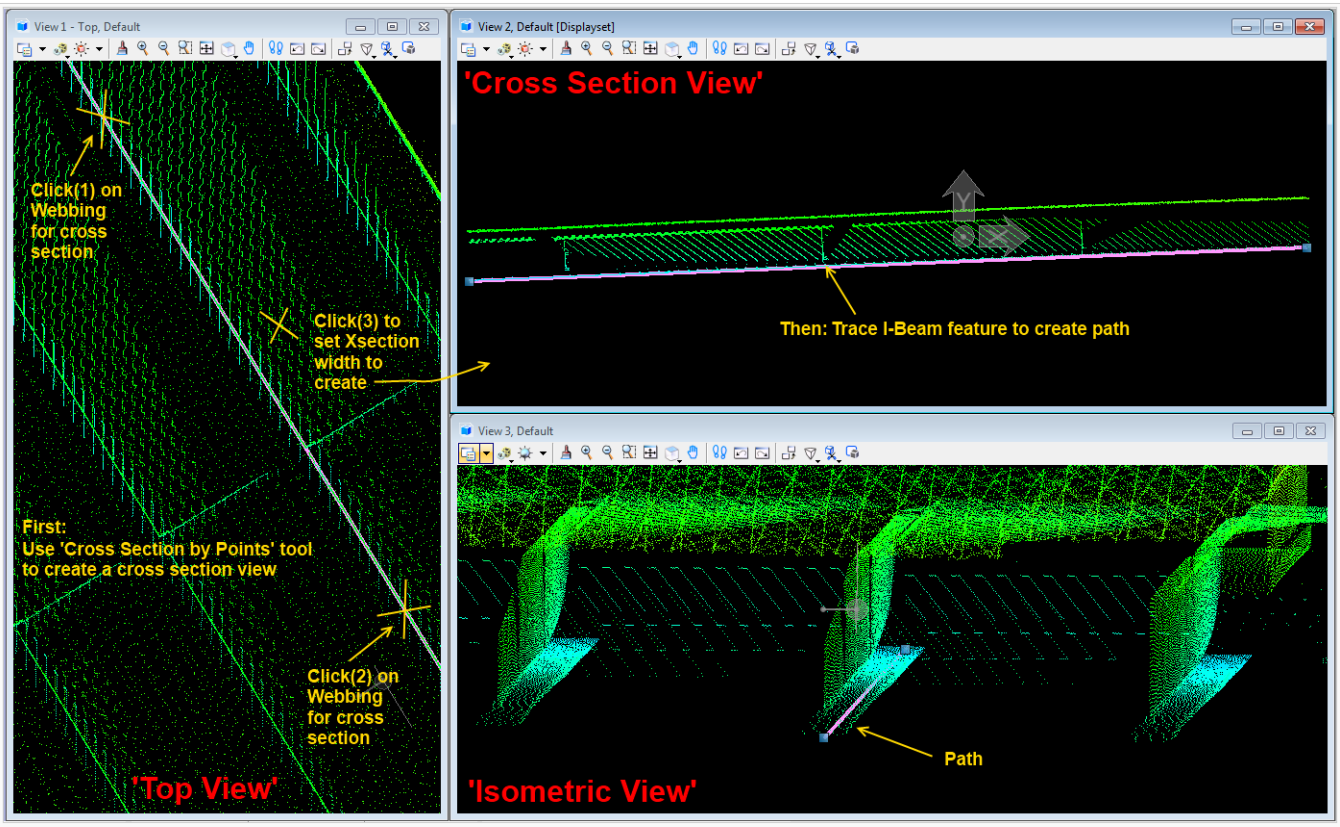
Tool Settings

Tool Settings	Effect
'd'	Beam Height
'f'	Flange Thickness
'b'	Flange Width
'w'	Web Thickness
Place	Place the specified beam as a CAD Element

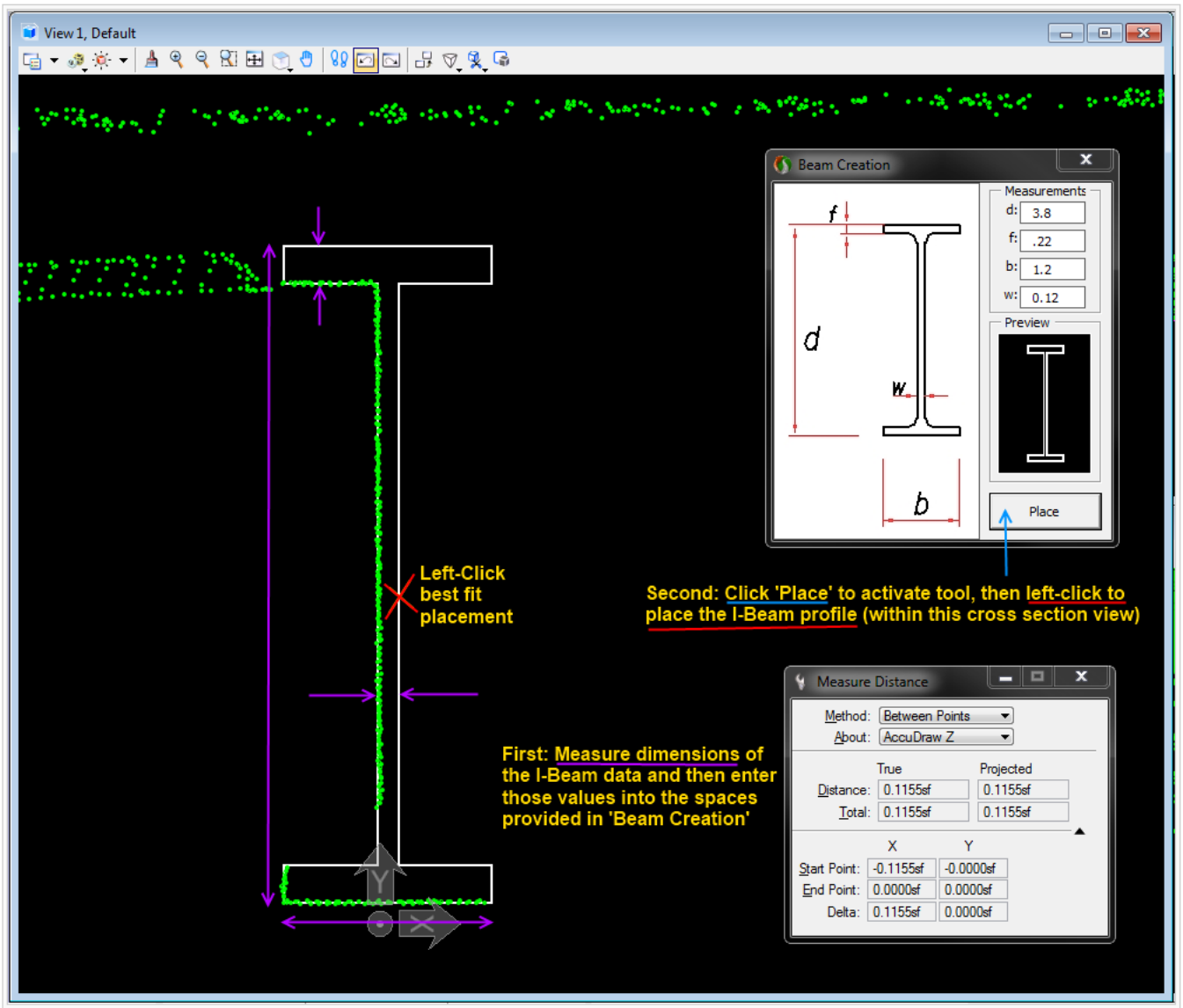
How To's

To Create Beams based on measured Dimensions

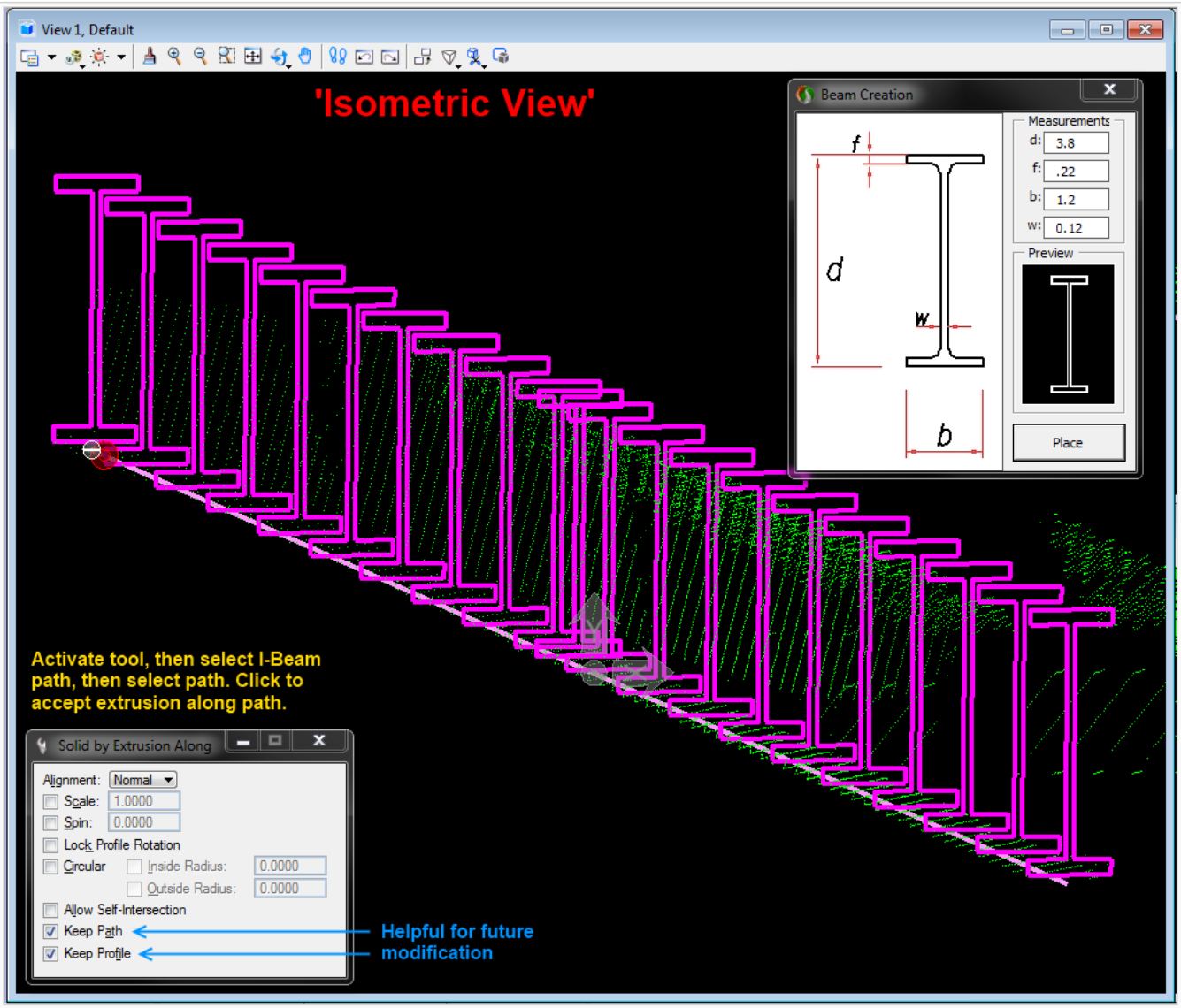
1. Isolate or locate beam data and have multiple perspective views ready for reference.
2. From 'top view' take a cross section down a visible edge (usually the web face) of the I-Beam, see Cross-section by Points.
3. From the 'cross section view', draw a path down the I-Beam (a profile will follow this path later).
4. Now establish another cross section along the newly drawn path (such that a profile is best visible in the data).
5. Measure the dimensions of the I-Beam (within the 'profile' cross section) using MicroStation's 'Measure Distance' feature. See Help/Contents/Search(tab)/'type measure distance'.
6. Activate I-Beam tool by clicking icon. Record measured values in spaces provided.
7. Click 'Place' to place a CAD profile within the 'profile' cross section. Place Profile.
   - To extrude profile, use MicroStation's 'Solid by Extrusion Along'.
   - Check 'Keep Path' and 'Keep Profile'. First select path, then select profile, and left-click to accept.
   - The profile has been extruded along the path
8. Use this profile for other beams or restart process for new ones.



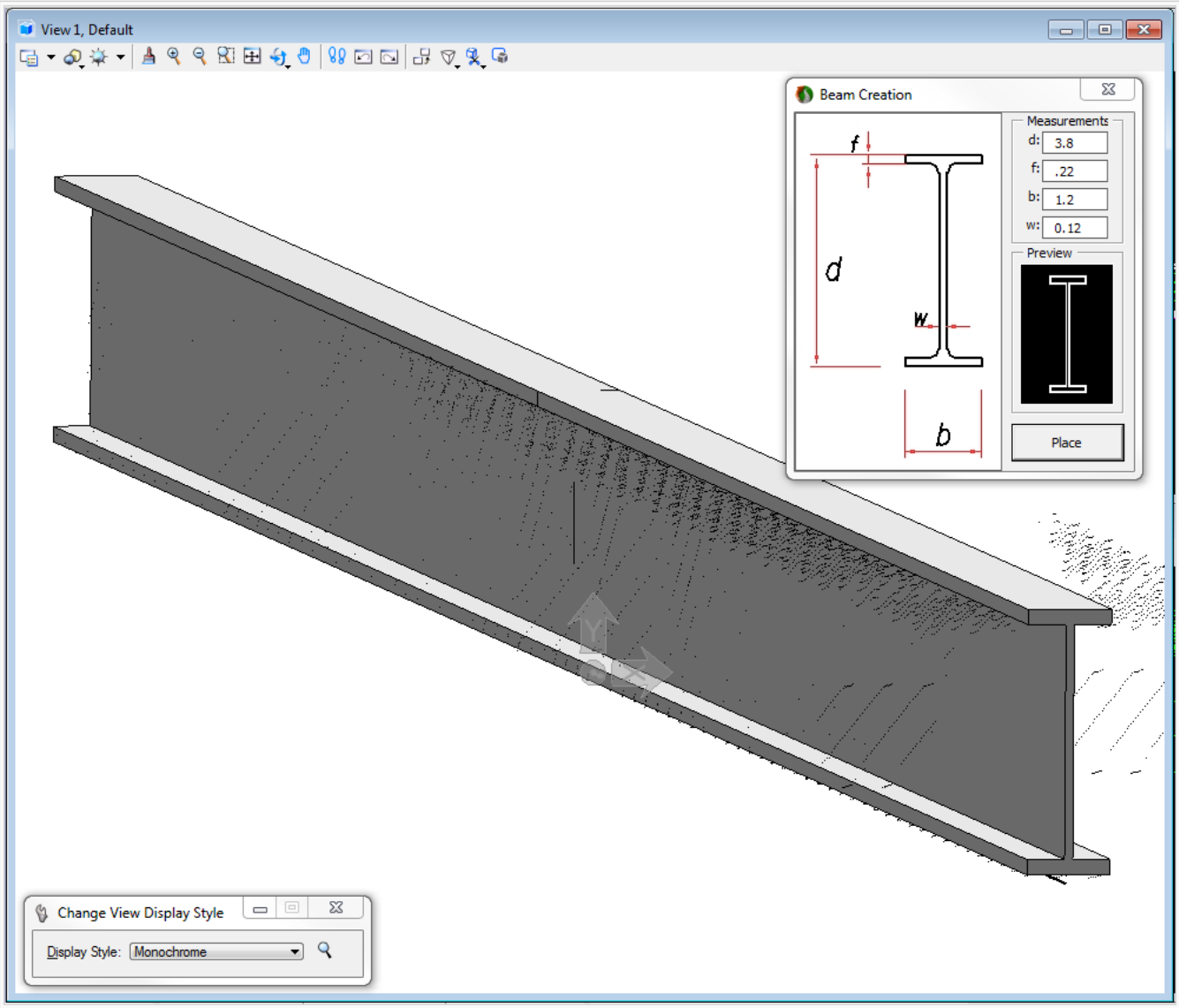
Window for setting up cross section and path.



Window for setting up an I-Beam profile.



Window for setting up an extrusion of the profile along the path.



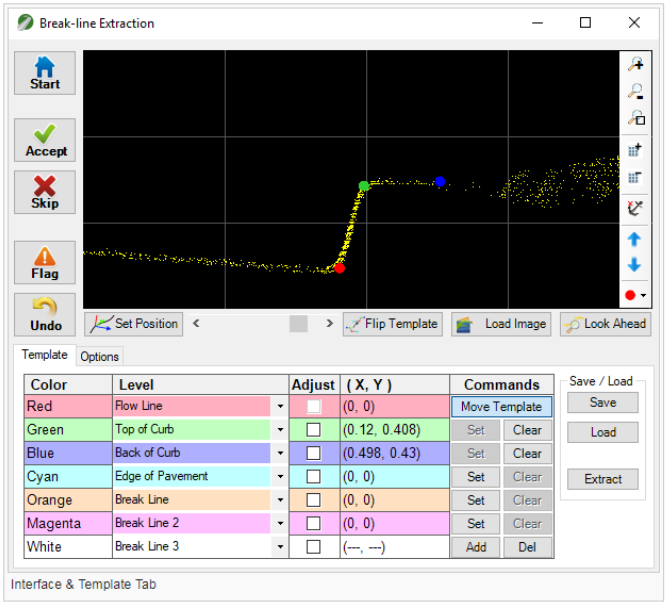
Window showing results of the extrusion.

Note: Before placing a profile in the 'cross section view', make sure to measure within this view beforehand (in-order to activate the view for placement). If the profile does now match data (such as, rotation or size), the CAD element can be modified by dragging/moving nodes/rotate. Also, I-Beam paths may need to be adjusted by adding more nodes (to fit any deflection/curvature).

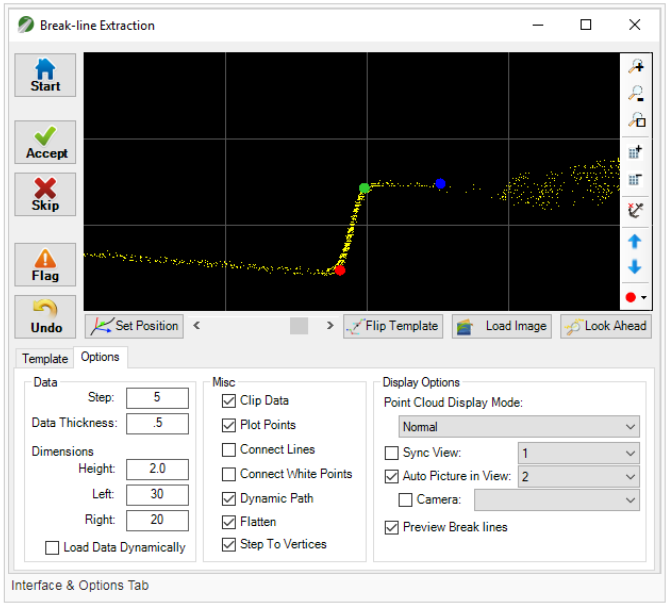
Break-Line Extraction

Purpose

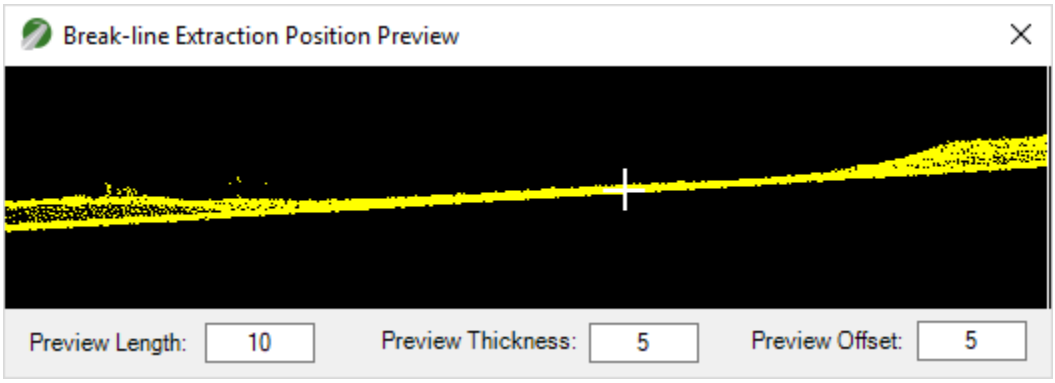
The Break-line Extraction tool improves the accuracy of placing curbs & other break-lines based on a user determined profile. This tool will follow a user defined path to display an enlarged cross section of the feature to identify its shape, detail, and complexity. The blown-up section width is defined by the user. For the tool to be applied in sections larger or smaller than a curb. The user defined path can be as intricate as necessary; this tool is not restricted by the use of curves or splines.



Interface & Template Tab



Interface & Options Tab

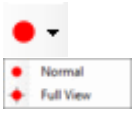


Tool Settings

Break Line Extraction Interface

	Tool Settings	Effect
Interface	Start	Sets the currently selected element as a guide line for breakline extraction. This will also reset to a new series of extraction if you are currently mid extraction.

Accept	Accepts and plots the current position and placement of the Template Points. After 2 'Accepts' you should see poly lines form. Hotkey (Enter)
Skip	Skip the current position on the guideline. Hotkey (')
Flag	Flags the current position on the guideline. Hotkey (;)
Undo	Undo any previously accepted Template Points.
Set Position	Allows you to move to a new position on the guide line. Simply activate feature and snap to any location along your guide line.
Position Slider	Make micro-adjustments for the positioning of the tool along its path/guide-line. Click arrows for 1 unit intervals or click slider itself for x>1 unit intervals.
Flip Template	Flips/mirrors the Template Points about a vertical line.
Load Image	Automatically load the closest image to the working area.
Look Ahead	Enable this to display your Position Preview which is a cross section view parallel to your guide line. This can be used to help identify where curbs terminate or transition into ramps/driveways. *Note: If the guideline is on the face of the curb and offset is 0' and thickness is 0.5' then you'll have a good view at the face of curb to better identify transitions.*
 Zoom In	Zoom in the preview window.
 Zoom Out	Zoom out the preview window.
 Zoom Reset	Reset the preview window.
 Increase Grid Block	Increase grid blocks in the preview window.
 Decrease Grid Block	Decrease grid blocks in the preview window.
 Reset Anchors	Reset any anchors created between template items. *This essentially drops all the anchors set*

	 Vertical Scale	Used to Increase or Decrease the vertical scaling of the data within the preview. *Use to help identify faint breaks*
	 Cursor Toolbox	Cursor display type (Normal or Full View).

Template Tab

	Tool Settings	Effect
Template	Color	Color of template item when created in design file.
	Level	Level of template item when created in design file.
	Adjust	Automatically drop/shift template item vertically to the data
	Move Template	Move the template as one object.
	Set	Place template item within preview window
	Clear	Remove template item from preview window
	Save	Save current template as a .rcf file
	Load	Load a previously saved template (.rcf file)
	Extract	Copy a template from within the design file

Options Tab

•	Tool Settings	Effect
Data	Step	Stepping distance to move to the next position along guide line.
	Data Thickness	Thickness of point cloud data to process at each position.
	Height	Elevation distance of data from the guide line to display in the preview window.
	Left	Left distance of data from the guide line to display in the preview window.
	Right	Right distance of data from the guide line to display in the preview window.
	Load Data Dynamically	Loads relevant data inside the cross section and unloads any un-relevant data. *This is generally used where data has yet to be loaded on a feature.*
Misc.	Clip Data	Clip point cloud data at each position.
	Plot Points	Place point elements at break points.

	Connect Lines	Connect break points with a linear element.
	Connect White Points	Connect white points.
	Dynamic Path	Continue a path after the end of the guideline.
	Flatten	Temporarily flattens the guideline so that 'up' in the cross-section preview is perfectly vertical. No matter the tilt of the guideline, this will show you a plumb plane. *This is not always necessary because an ideal preview is one that is perpendicular to that of the features profile.*
	Step to Vertices	Cross sections will automatically stop at every vertex
	Sync View	If enable, the current cross section will be centered in the view specified. This will essentially follow the ACS as it travels down the Guide Line. *There is now an option to SYNC ALL views within the drop down, which is usually done when there is no calibrated imagery.*
	Auto Picture in View	Automatically load the closest image to the current working area
	Camera	When checked, will use only the selected camera for image loading.
Best Fit Options	Angle Deviation Limit	Error Tolerance allowed when using the 'Auto/Auto-Fit' feature.
Display Options	Preview	When checked, enables live preview of template placement. Disabling improves tool performance.
	Display Mode	Drop down menu for the different display modes available. This will change the display mode within the Tool Preview ONLY!

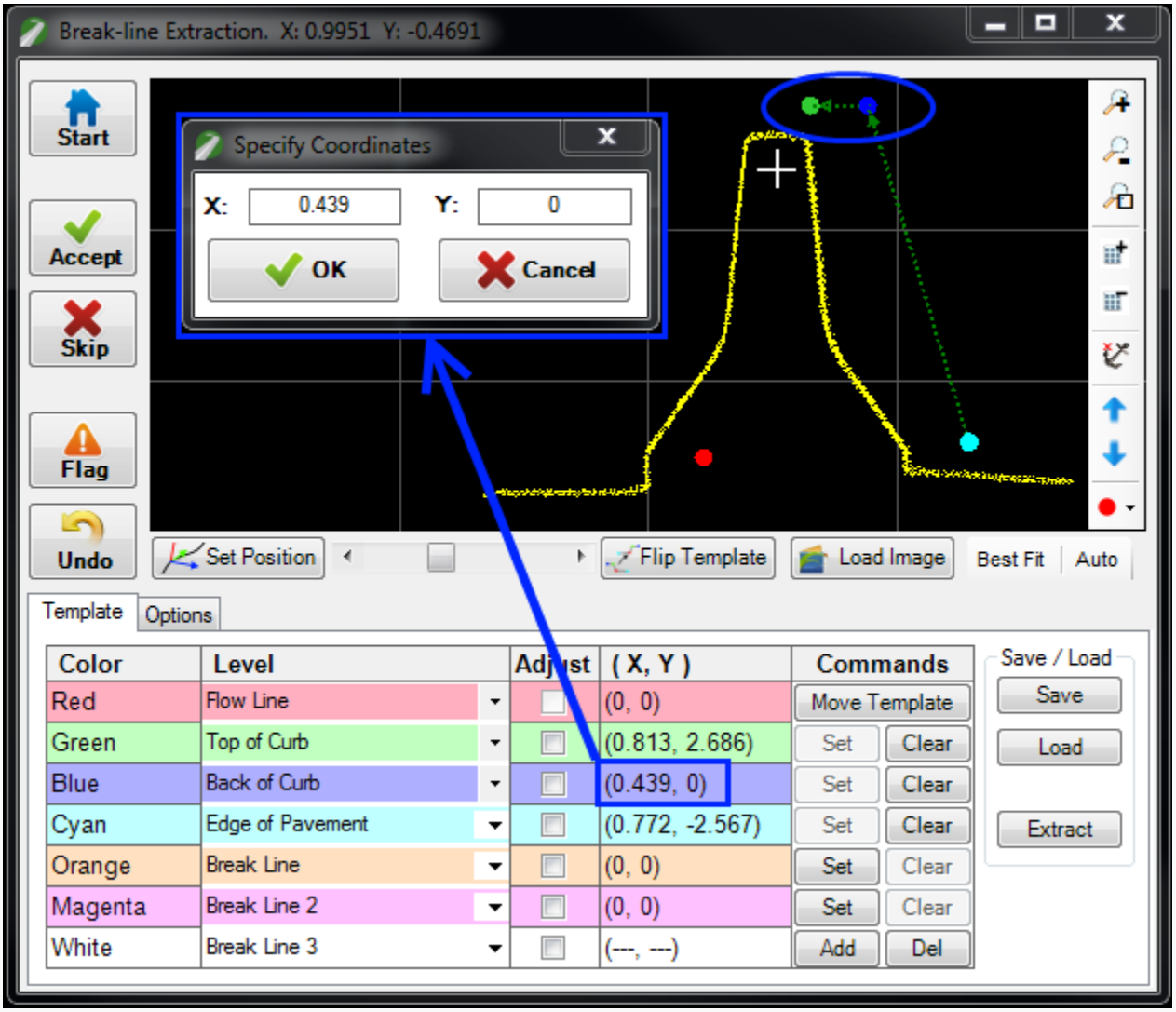
Note: Once a template point has been 'Set' it is now possible to manually enter custom (x,y) coordinates for that point. If the point is not anchored to another then the relative (0,0) position will be the Red Point. The (0,0) position for an anchored point will be the point in which it is anchored too. For example, if the Blue point is anchored and only 1 unit right to the Green point then the relative coordinates of the Blue point will be (1,0). Hint: Double click on the specific color's (x,y) to modify.

Note: Sync view can be applied to all views and will not interfere with 'Auto Picture' feature.

Note: Tool will accept a template file path as a key-in parameter. Ex: "TopoDOTApp Breakline P:\Projects\Route 7\template.rcf"

Note-hotkey: Hold SHIFT to restrict movement to vertical, Hold CONTROL to restrict movement to horizontal. This applies to the movement of the template or a specific template point.

Note-hotkey: Use the bracket "[" and "]" hotkeys for moving the viewpoint to the next/previous available template point. This will help significantly if a template (or feature/profile) has relatively large dimensions, such as an entire highway or bridge profile.



Adjusting Template coordinates when anchored

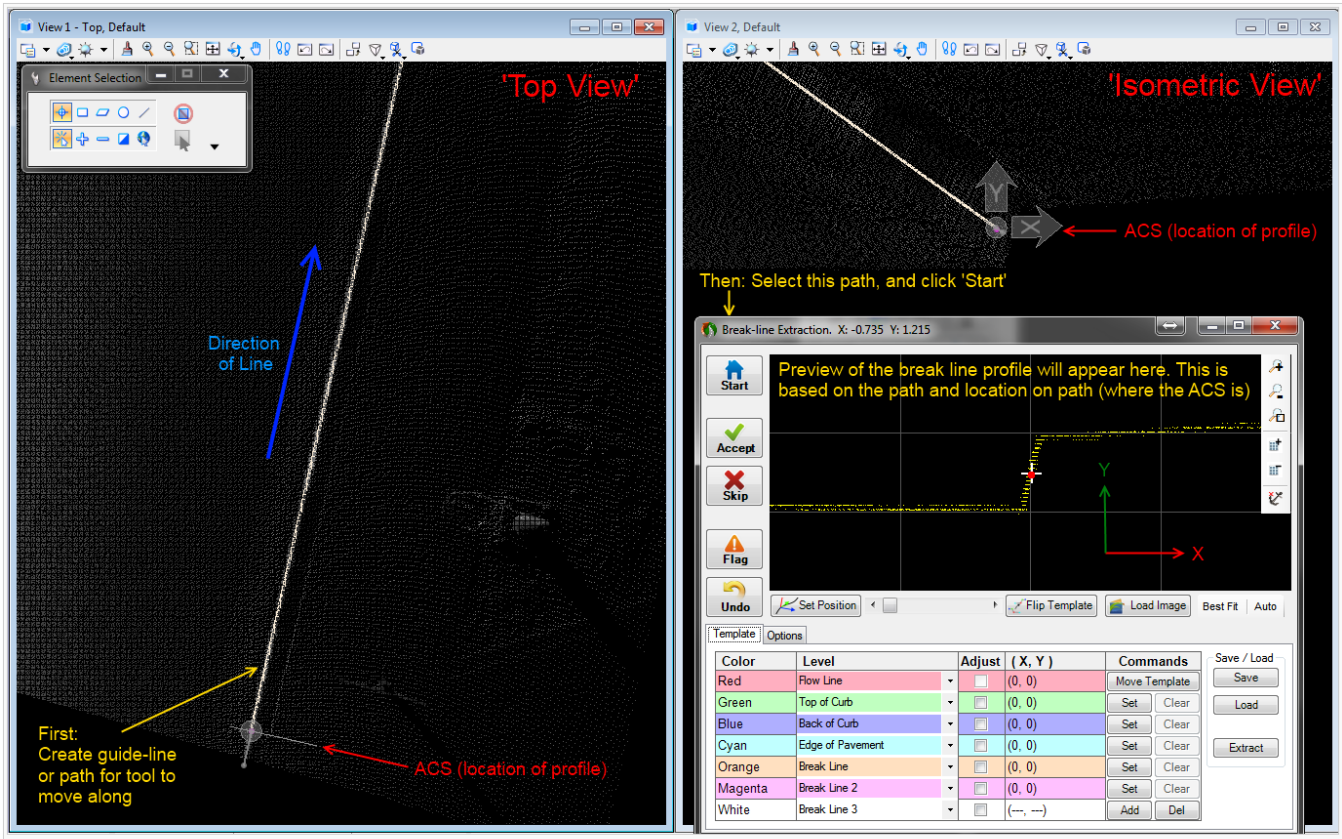
How To's

To Extract a Break Line

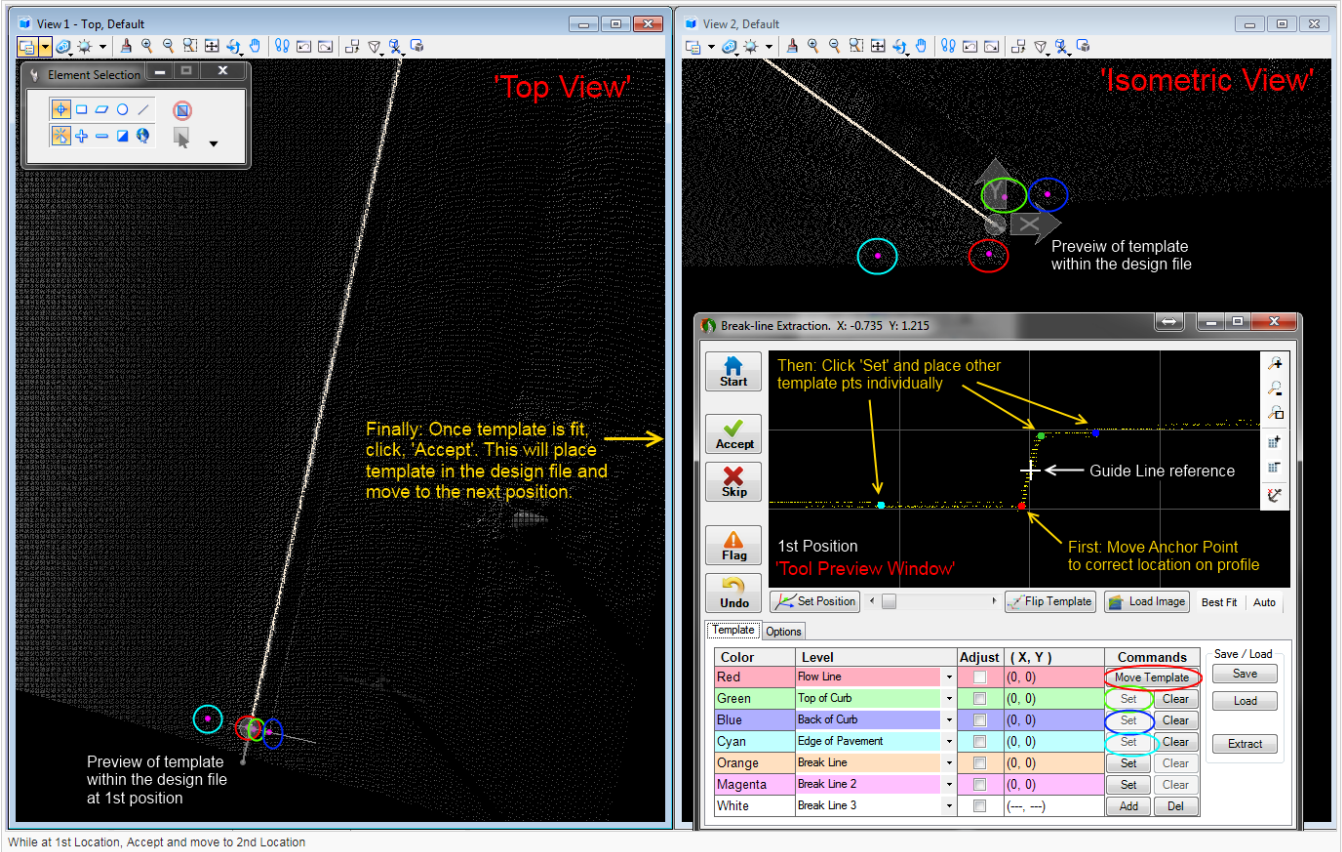
1. Isolate a region of data which includes the break line to be extracted.
2. Place a quick poly-line along the break line. This line will be the path or guide-line for the tool to run along. *A quick way to place a line is using Point Snap*
   - Make sure the path is parallel and close enough (Dimension settings relates to path) to the break line.
   - As you progress you will notice that you can also use a common centerline such as a paint stripe for most the project. This is useful when working with highway corridors or long straights of road with uniform profiles.
3. Open the Break-Line Extraction tool.
   - Depending on which type of break line (curb, barrier, wall) is being extracted, adjust settings within the 'Options' tab to match this type of feature.
   - Enable 'Connect Lines' and 'Draw Points' to add lines and points to the design file while extracting. *Recommended*
   - Adjust Data Dimensions to encompass the entire profile of the break line. *Optional*

- Disable 'Clip Data' to avoid clipping while tool moves down path. *Optional*
- *Read other options within the Tool Settings to get familiar with all features.*

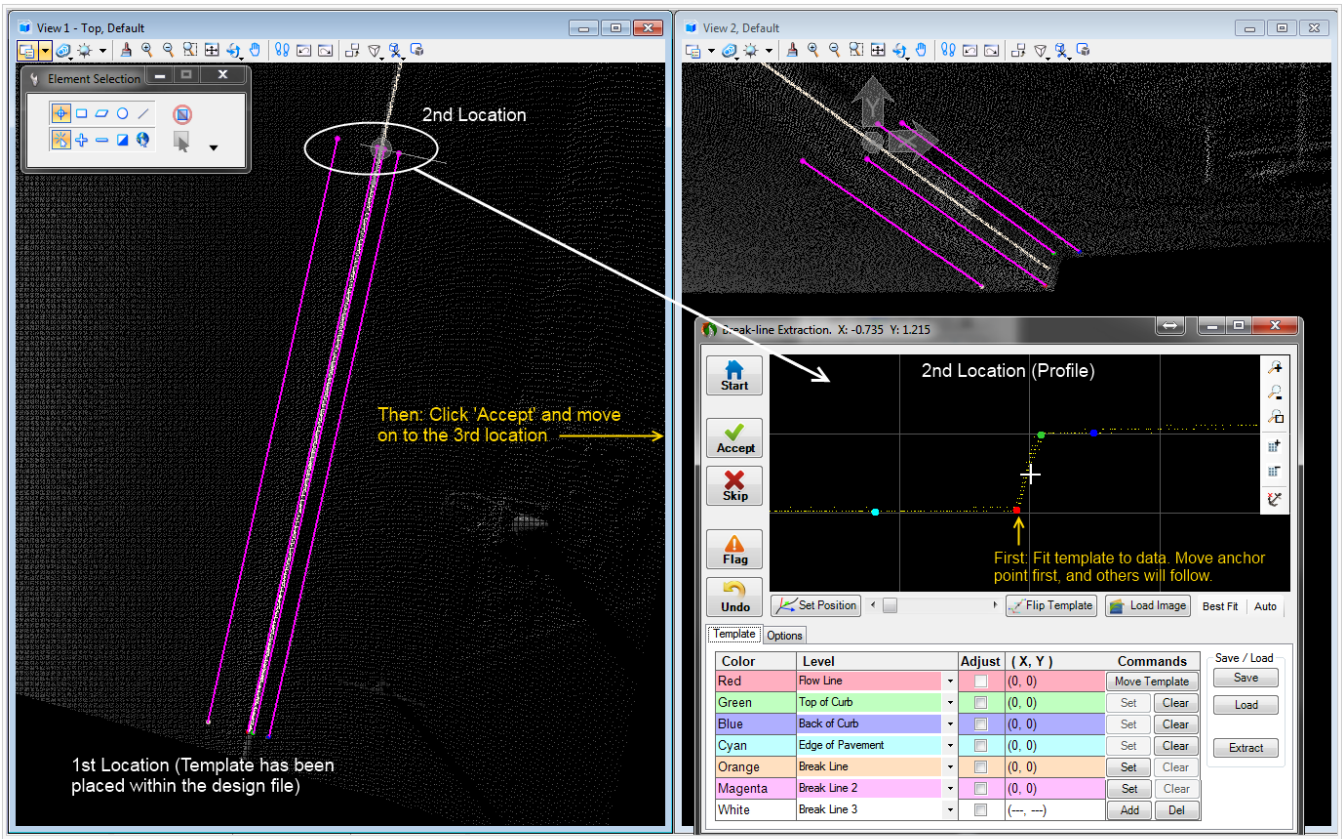
4. Select guide-line, and then click 'Start'. This action sets the tool along the path. This position will be at the beginning of the path.
   - To get a better preview, the position may be moved to a different location where data is better. Do this by clicking 'Set Position' and clicking the new location/position along the guide-line.
5. Now establish a template to fit the data. *These template items will be placed in the tool preview where there is a cross section of the break line.*
   - Click 'Move Template' and place the first point within the preview. *This point will be the anchor point, in which all other points are slave to its movements*
   - Click 'Set' for the next template item, and place within the preview.
   - Continue to place other template points by moving down the list: clicking 'Set' and placing in the preview.
   - Adjust the 'Level' name to the corresponding template point.
   - Now that the template is created, click 'Save' to create a .rcf file for future use. *Optional*
6. It's time to start extraction.
7. Click 'Set Position' to move tool to a starting location, or simply click 'Start' to move to the beginning of the guide-line. *This will display a new cross section of the break line within the tool preview*
8. Within the tool preview window, click on the anchor point (Red template point) and now move the template. Move the template to the data and match it to the profile of the break line.
   - The template may not fit perfectly, so if adjustments need to be made then click on another template point and move to data. *This movement is done within the preview window of the tool. Optional*
     1. While moving a point, the user can hold 'Shift' to lock any movement to a vertical direction (OR) hold 'Control' to lock movement to a horizontal direction. *Optional*
     2. Use a combination of the Zoom, Grid, and Anchor settings while placing these points. *Optional*
9. Now that the template is fit for the first position, click 'Accept' to place these template points within the design file.
   - The tool automatically moves to the next location (at the specified 'Step' value) and gives you a new cross section of the break line.
     1. If preview is bad, move position by clicking 'Set Position' and click on guide-line or using the 'Slider Bar'. *Optional*
     2. User can 'Skip, Flag, or Undo' at this point. *Optional*
   - Continue to move template in-order to match the data profile and accepting
10. If the tool is closed or restarted or started on another guide line, this will create a new extraction process; therefore, there will be no more interaction between the new break line and first break line.



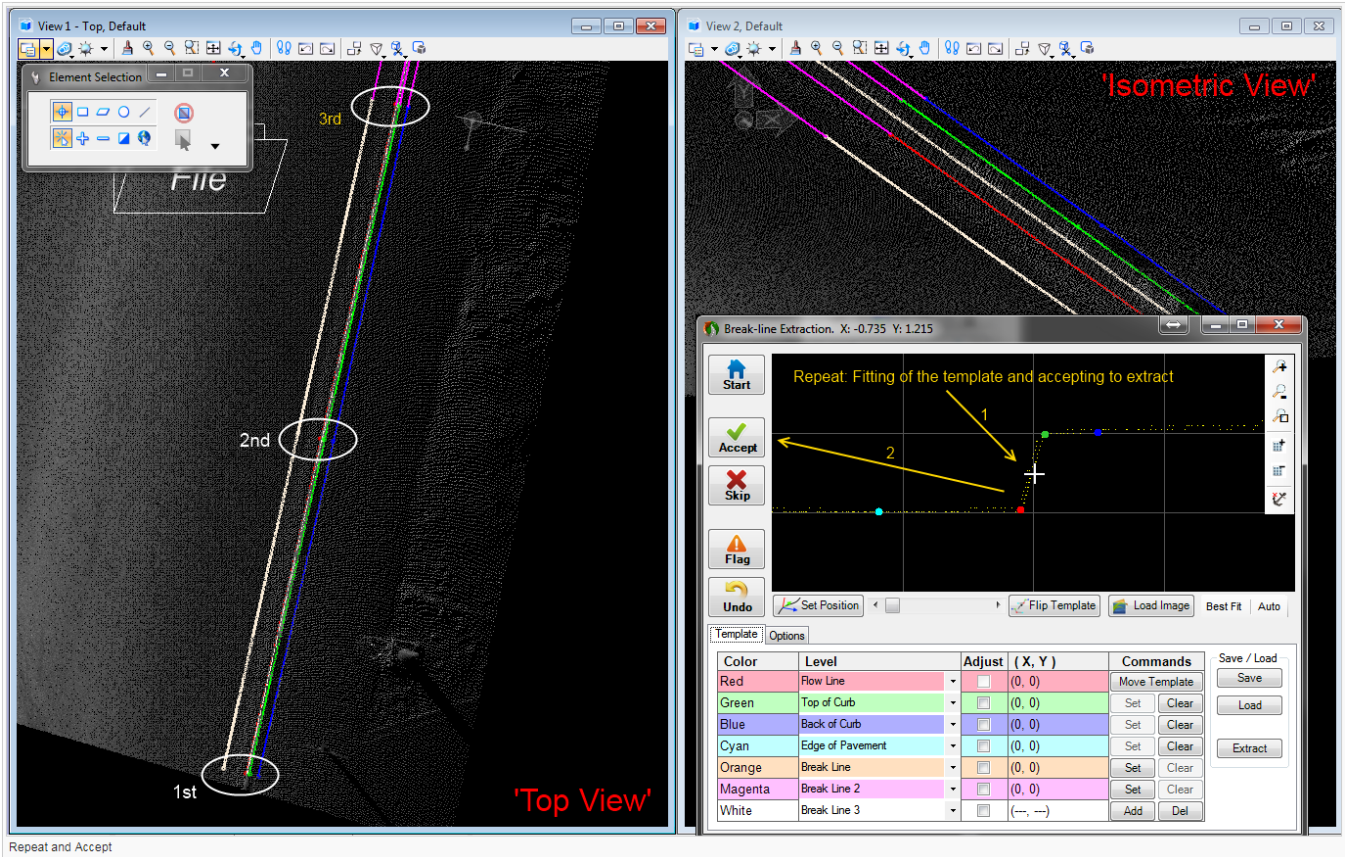
Setting Up Template



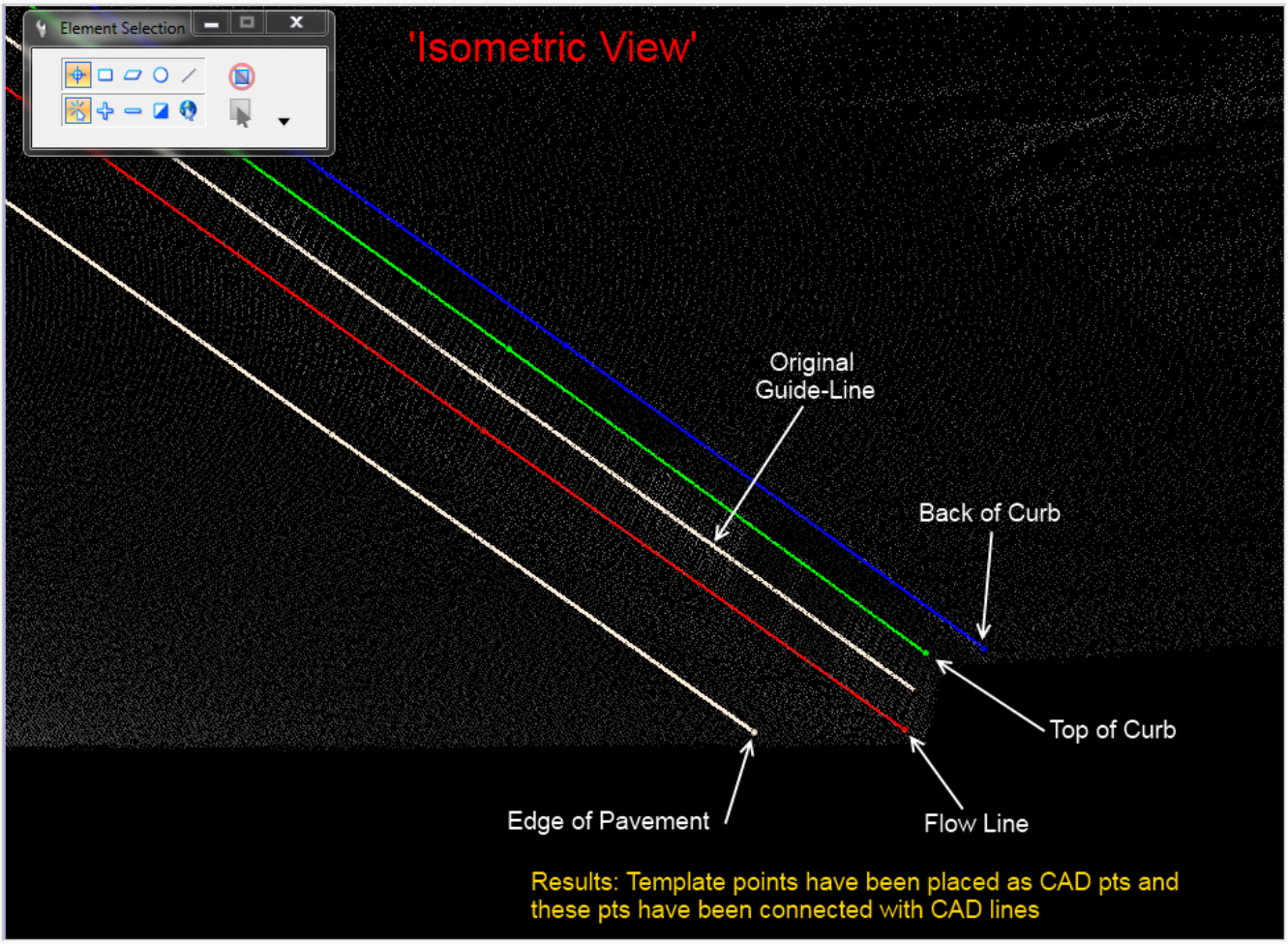
While at 1st Location, Accept and move to 2nd Location



Now at 2nd Location, Repeat and Accept



Repeat and Accept

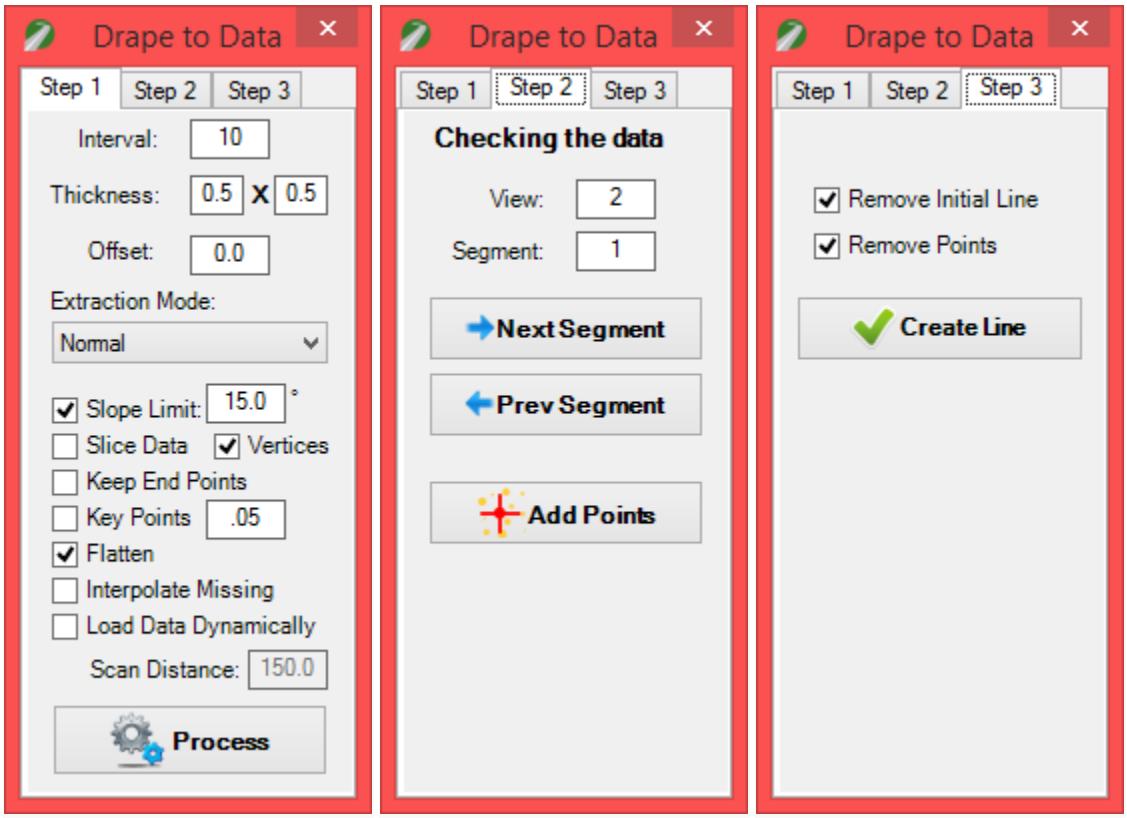


Results of the Break Line Extraction

Drape Element to Data

Purpose

Drapes a polyline or shape element down to the point-cloud-data elevation at specified intervals or at defined vertices/handles. Missing data (such as shadowing around a building) can also be accounted for with the interpolation feature of this tool.



Drape to Data Step 1 | Drape to Data Step 2 | Drape to Data Step 3

NOTE: While operating on a single line, the tool must be used from step one to step three. 1) Is to specify options and execute, 2) Is to QAQC line segments, 3) Is to apply results.
NOTE: While operating on multiple lines, the tool will automatically apply results when 'Process' has been clicked; therefore, there will not be a step 2 or 3 involved when processing multiple lines at once.
NOTE: 'Add Points' within step 2 can be used when: the draping requires an extra node, there is missing data, the data is too complex/noise.
NOTE: Tool will now properly drape shape elements so that it retains the shape element type (shape will remain closed).

Tool Settings

Step 1

Tool Settings	Effect
Interval	Spacing between processed points
Thickness	Thickness of data to look at for calculating the ground level. This will be width and length along selected line.

Offset	Offset the data to look to the left or right. The X-axis of the ACS is a positive value. Will show the preview sample on the start and end of the element being processed.
Extraction Mode	Change extraction method: Normal (averages the whole sample), center-weighted (data within center of sample have greater effect on the average), low point, or high point for the data sampling method.
Slope Limit	Removes isolated points that exceed the specified slope. This slope is based on a horizontally flat reference.
Slice Data	Takes a cross-section or slice at every interval. If enabled, this will override any cropped point cloud information.
Vertices	Forces tool to calculate an elevation at all vertices of the selected element.
Keep End Points	When enabled, the start and end points of the element are not moved. *Does not modify the ends of the selected poly-line or shape. You'd do so if the beginning and end of your line tie into existing line work that is already correct (in elevation and horizontal).*
Key Points	Plots key points so that straight/flat sections receive a low amount of points and sections of change receive a higher amount of points. The text box is for the XY&Z tolerance values. The tool will point(s) once it deviates out of the specified tolerance.
Flatten	When enabled, measurements will be taken vertically instead of perpendicularly to the reference line. So, if 'Flattened' then the sample will be projected down vertically to the data. Recommended to do this when working with a tilted line. If the line is already flat then there will be no difference between enabled or disabled.
Interpolate Missing	Interpolates an elevation on vertices/handles/intervals with missing data. The linear-interpolation will use available neighboring point elements to calculate the elevation on points with missing data.
Load Data Dynamically	Loads relevant data inside the cross section and unloads any un-relevant data. *This is generally used where data has yet to be loaded on a feature.*
Scan Distance	Determines the relevance of scan data based on the specified distance from the scan position to the cross section.
Process	Activates the tool and processes the selected line.

Step 2

Tool Settings	Effect
View	The desired view used to display the current cross section or ACS orientation.
Segment	A segment is a region from one handle/node to the next on a poly-line. The value displays the current segment being examined. *The current segment will be highlighted within the design file.*
Next/Previous Segment	Navigate to the next/previous segment. *This feature will NOT work on B-Splines and a manual check will be required.*
Add Point	Allows the user to insert a Temporary point for the element/polyline to be draped to. This is essentially like manually inserting drape points one at a time for the current segment.

Step 3

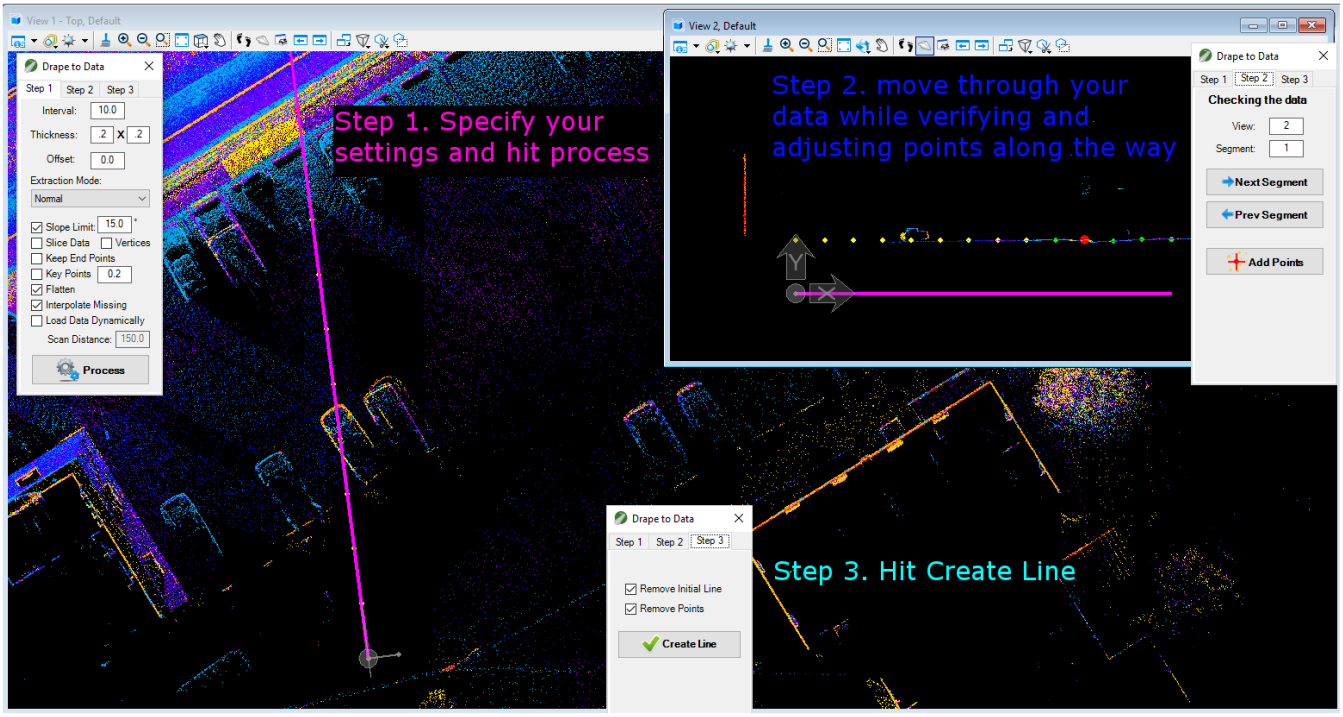
Tool Settings	Effect
Remove Initial Line	Will remove the original/initial line once draping is complete. *In its place, a newly draped line will be created.*
Remove Points	Will remove the Temporary Points used for visual inspection. *These are the green, red and blue points. Keep these if you need point elements instead*
Create Line	Activates the creation of a newly draped line and will carry out any enabled features within Step 3.

How To's

http://www.certainty3d.com/university/training/beginner/B106/

To Drape an Element

1. Open the 'Drape Element to Data' tool
2. Specify the settings that you wish to use prior to running the tool.
3. Select the line that you wish to drape. Select 'Process'.
   - *This tool can now properly drape shape elements. The shape will be draped while remaining closed.*
4. With the line still selected, move to the 'Step 2' tab. This is the QA/QC phase of the tool.
5. Specify which view you wish to see the cross section in from the 'View' input box.
6. Click on 'Next Segment' and examine the cross section view and verify that points have been placed correctly. Simply add, move or delete points as needed. *This feature of moving down segments will not work on B-Spline elements.*
   - Green points signify that points were placed with no error.
   - Red points signify that there was noise or lack of data involved in the elevation calculation.
   - Yellow points signify that there was a large deviation in the points' elevation.
7. With the point placement verified, move on to the 'Step 3' tab.
8. Select the options you wish to use and select 'Create line'. This will create a new line with a node at every elevation point calculated from the tool.

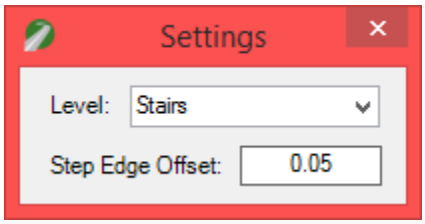

Step 1. Specify your settings and hit process
Step 2. move through your data while verifying and adjusting points along the way
Step 3. Hit Create Line
Drape to Data
Step 1
Step 2
Step 3
Interval:
Thickness:
Offset:
Extraction Mode:
Normal
Slope Limit:
Slice Data
Vertices
Keep End Points
Key Points
Flatten
Interpolate Missing
Load Data Dynamically
Scan Distance:
Process
Checking the data
View:
Segment:
Next Segment
Prev Segment
Add Points
Remove Initial Line
Remove Points
Create Line

Step Extraction

Purpose

This tool is designed to generate 3D poly lines representing a set of steps (or staircase) in one direction only. The staircase can be square, rectangular or trapezoidal in form (from top view). The rise and run needs to be linear or maintain a constant slope. This will not support a curved or arched staircase or a multi-pitch staircase...yet :-)

Here is a quick **Video** on how to use the tool: Step Extraction Tutorial



Tool Interface

Tool Settings

Tool Settings	Effect
Level	Define what level to output the stair line-work to.
Step Edge Offset	This will be the horizontal distance between the back of the first step and the top of the next step. *The direction of this measurement is parallel with the direction of inclination.*

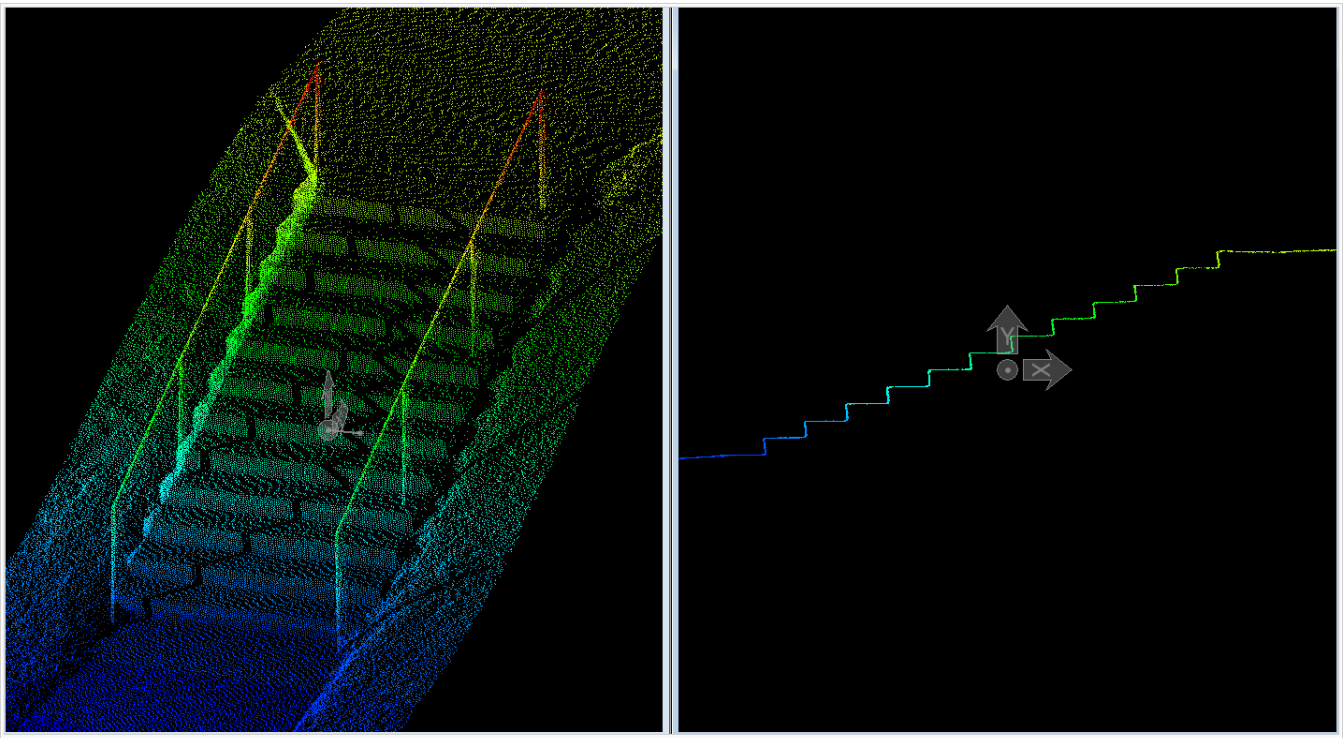
Note: If the stair poly lines are going to be used as a part of the DTM surface, makes sure the offset value stays positive and is larger than zero.

How To's

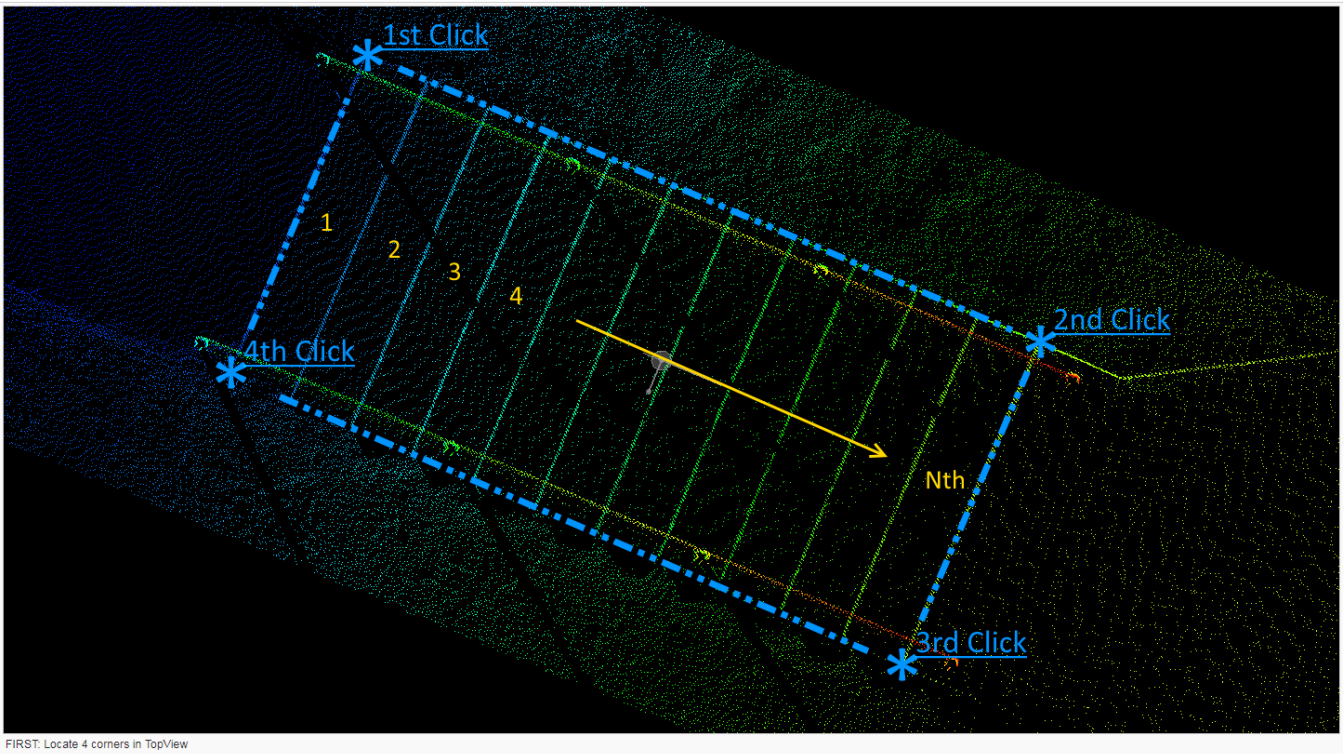
Here is a quick **Video** on how to use the tool: Step Extraction - Tutorial

To Generate a set of 3D Steps

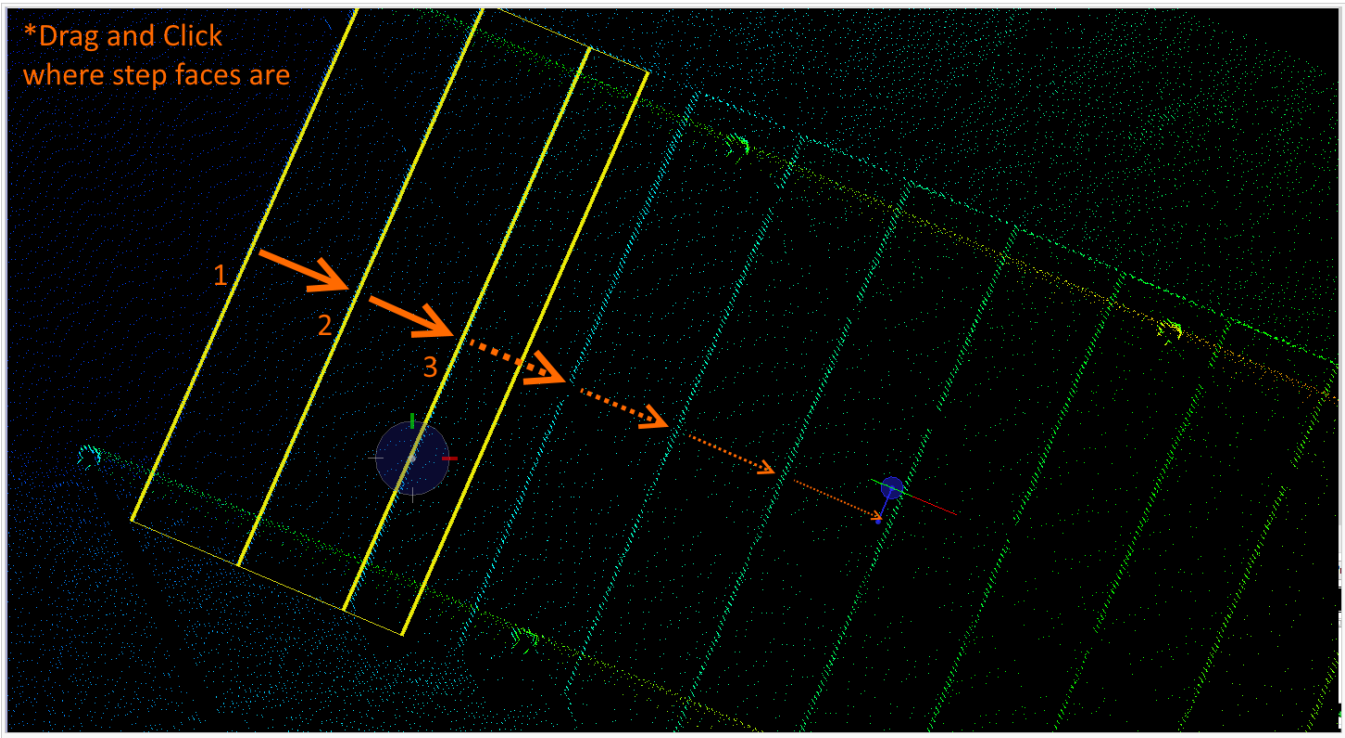
1. Isolate the stair case so you can clearly see from a top view. *Make sure the center of the staircase is clear of noise.*
2. Activate tool by clicking icon.
3. Click on all 4 corners of the staircase. Make sure that the 1st and 2nd click are on the side of the staircase (see image below).
   - Then move the cursor to the first step face and click to have the first step drawn.
   - Then move cursor to the second step face and click. Proceed to the next step until the end.
4. Once done locating faces of steps, right click to end the tool.



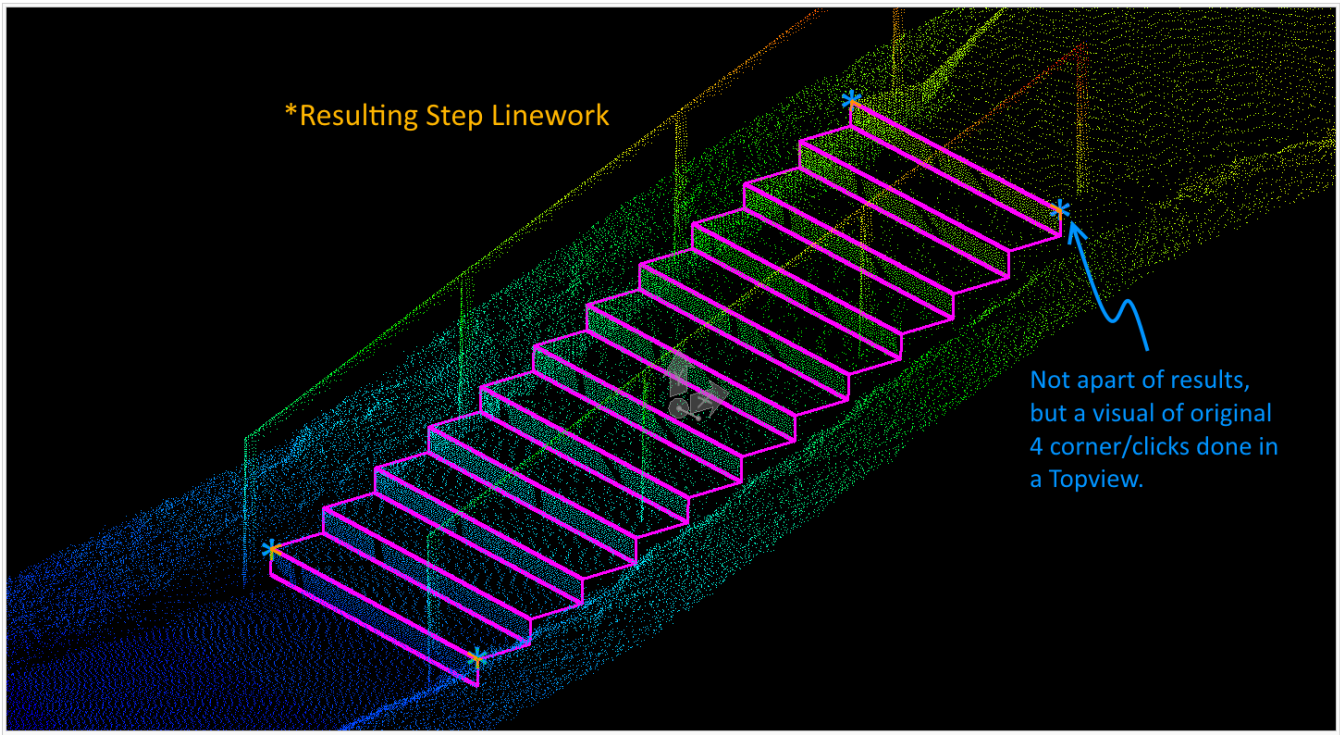
Typical Staircase



FIRST: Locate 4 corners in TopView



SECOND: Locate each step face



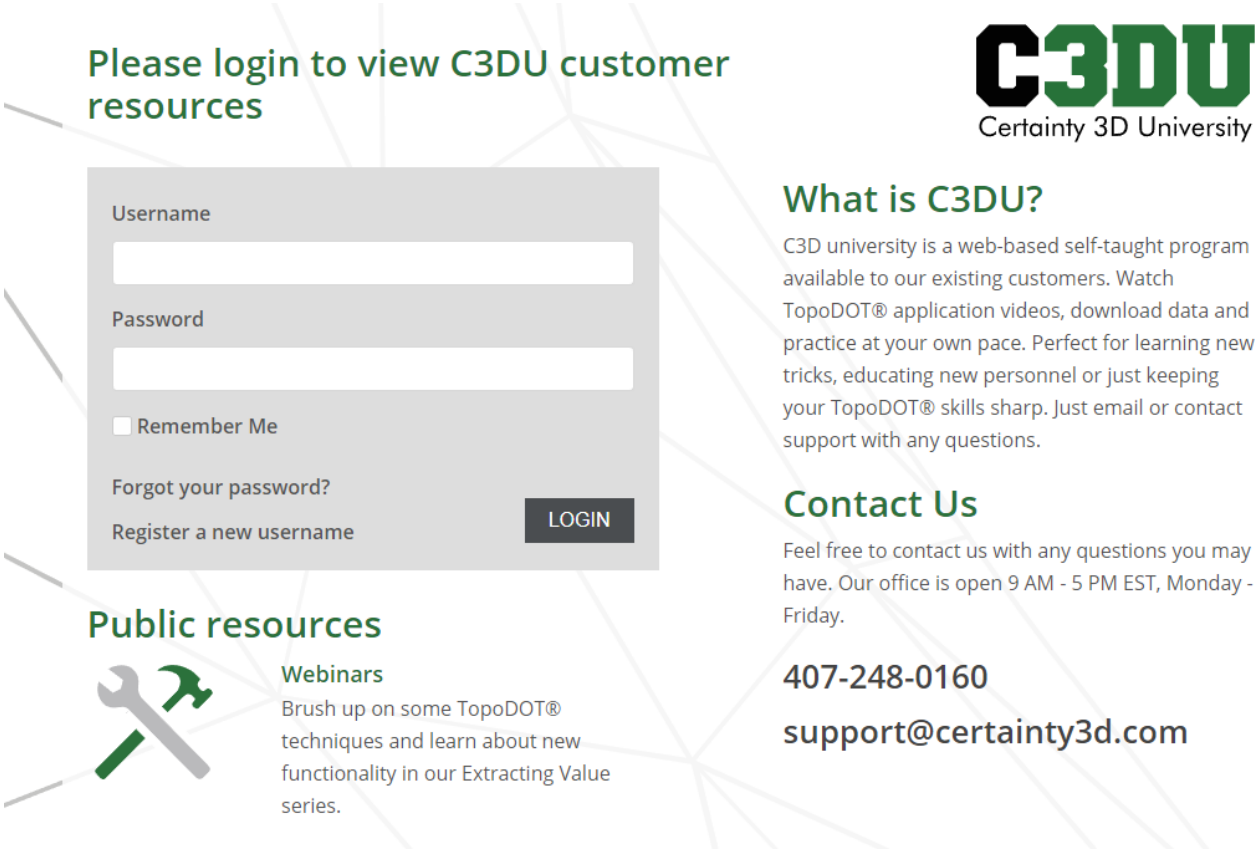
Results of tool

WIKI Page

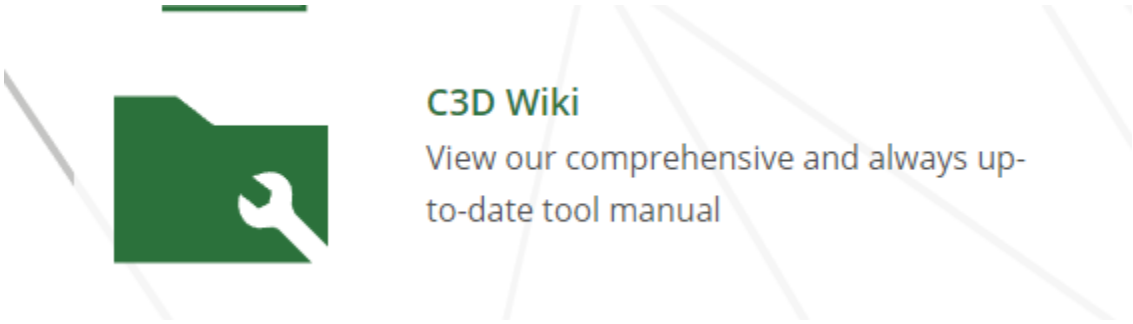
Steps to get on Wiki page

Step 1. http://certainty3d.com/university/

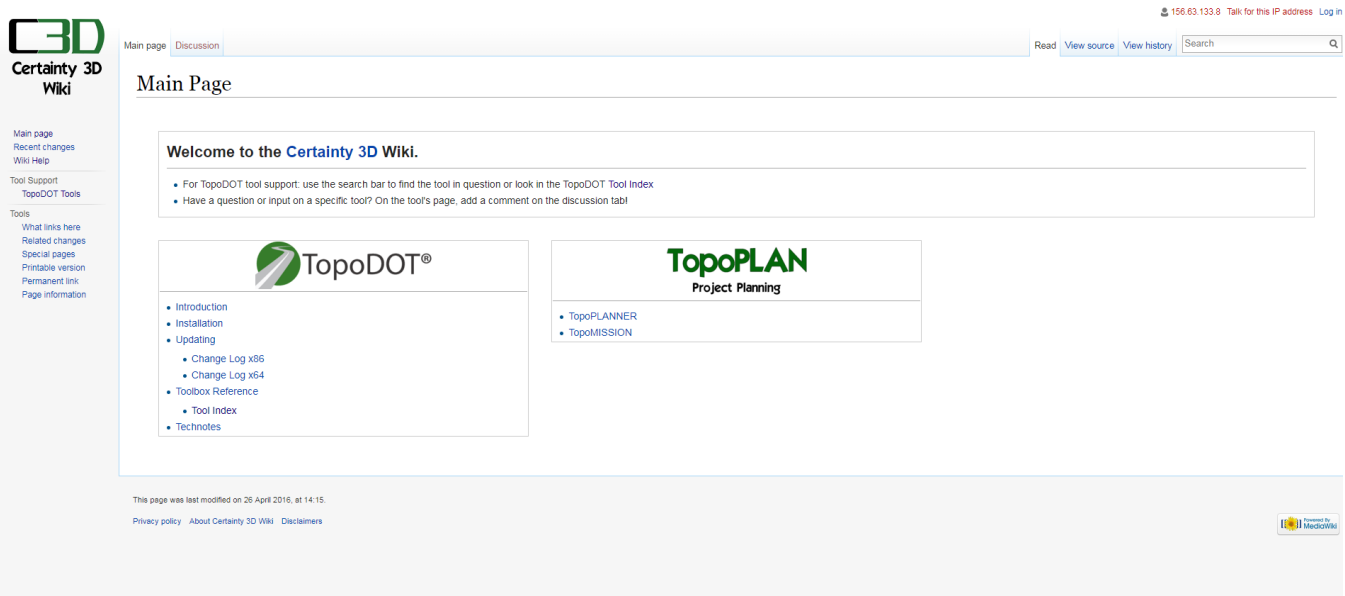
Step 2. Create a username and password



Step 3. Click on C3D Wiki



Step 4. This is the main page



Contacts

For any questions, suggestions, or problems with this document please contact the ODOT Office of CADD and Mapping Services by use of the following form on the ODOT website:

https://odot.formstack.com/forms/cadd_servicerequest