TerraScan Guide & Reference 1
Project & Data Management

REMS 6085,
Centre of Geographic Sciences, NSCC

Overview

This document is the first in a series of user guides designed to help you gain an understanding of Bentley's Microstation and TerraSolid's TerraScan software packages. The following topics are discussed in this guide:

Managing data in Microstation

- Creating Microstation design (DGN) projects
- Creating, modifying and displaying levels (i.e. layers)
- Creating and manipulating elements (i.e. features)
- Importing reference data into a DGN

Managing projects & blocks in TerraScan

- Opening the TerraScan MDL application
- Creating TerraScan projects
- Creating project blocks (manual & automatic)
- Importing LAS files into TerraScan project blocks

Working with data in TerraScan

- Querying project blocks
- Loading LiDAR points
- Querying LiDAR points

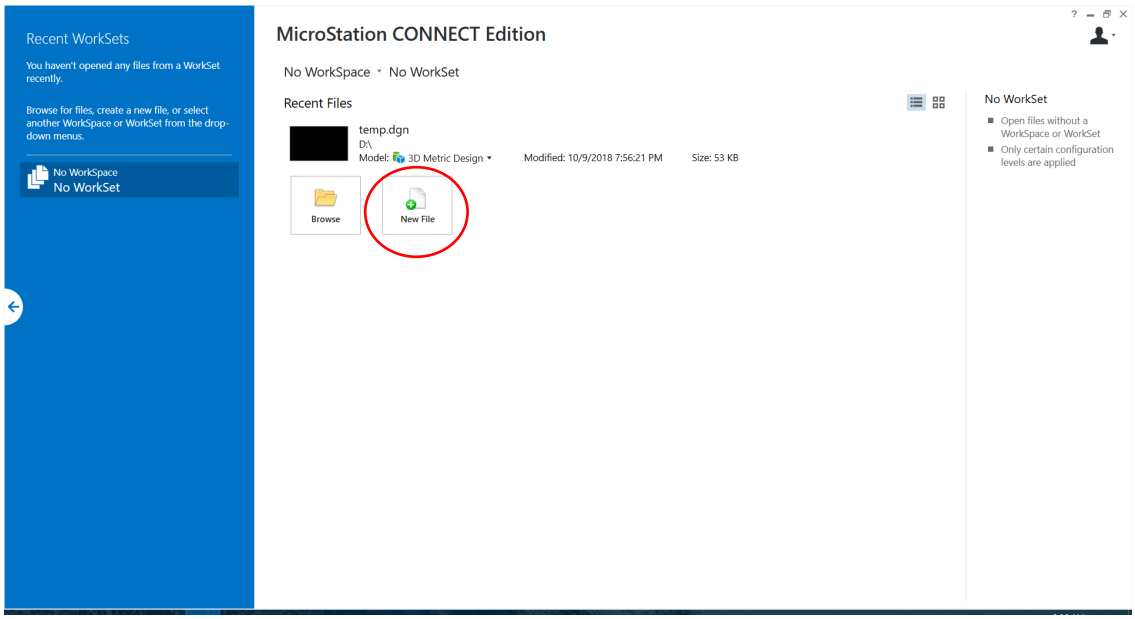
Note, that while the sections in this guide are partially organized in the order you may typically perform them, they are not intended as a rigorous step-by-step tutorial. Please **read through the entire guide before starting** as it will help your understand of the overall process.

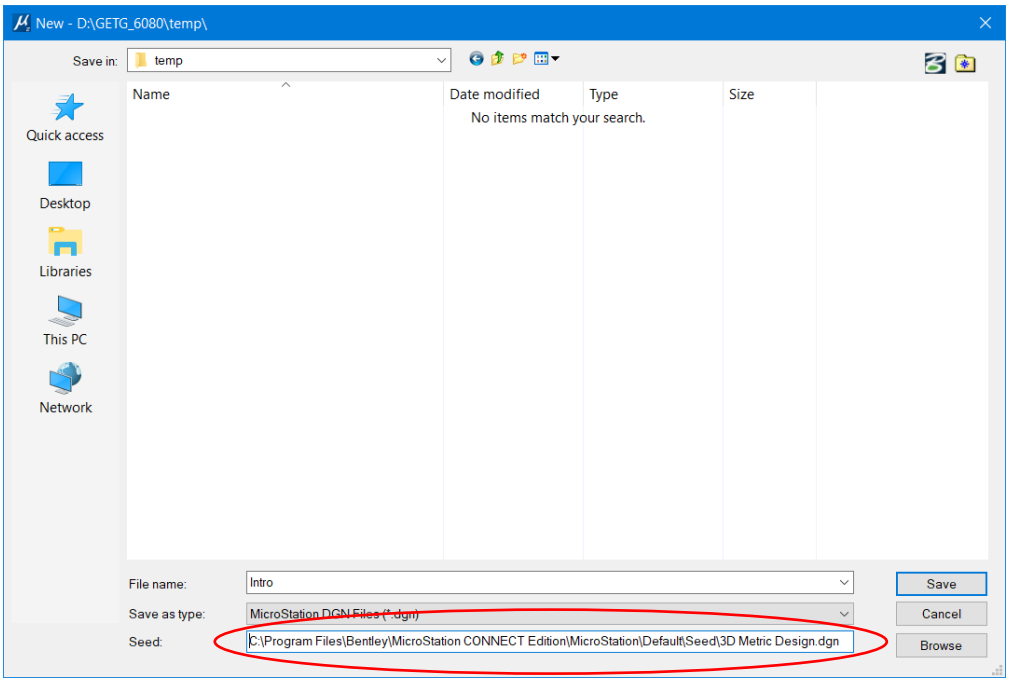
Managing Data in Microstation

TerraScan is not stand-alone software, but rather is an extension to Bentley Microstation - a popular CAD (Computer Aided Design) package. Many of the GIS-like operations such as creating and editing polygons are actually performed by Microstation. As such, the first few sections of this guide will introduce you to some of Microstation's core components.

Creating a Microstation DGN

When Microstation is started you are provided a window to select which DGN ("design") file you would like to load (below). In this case we are looking to create a new one, so select the *New file button* (circled below) to create a new empty DGN. Here you can specify the location and name of the DGN to create (bottom of page).



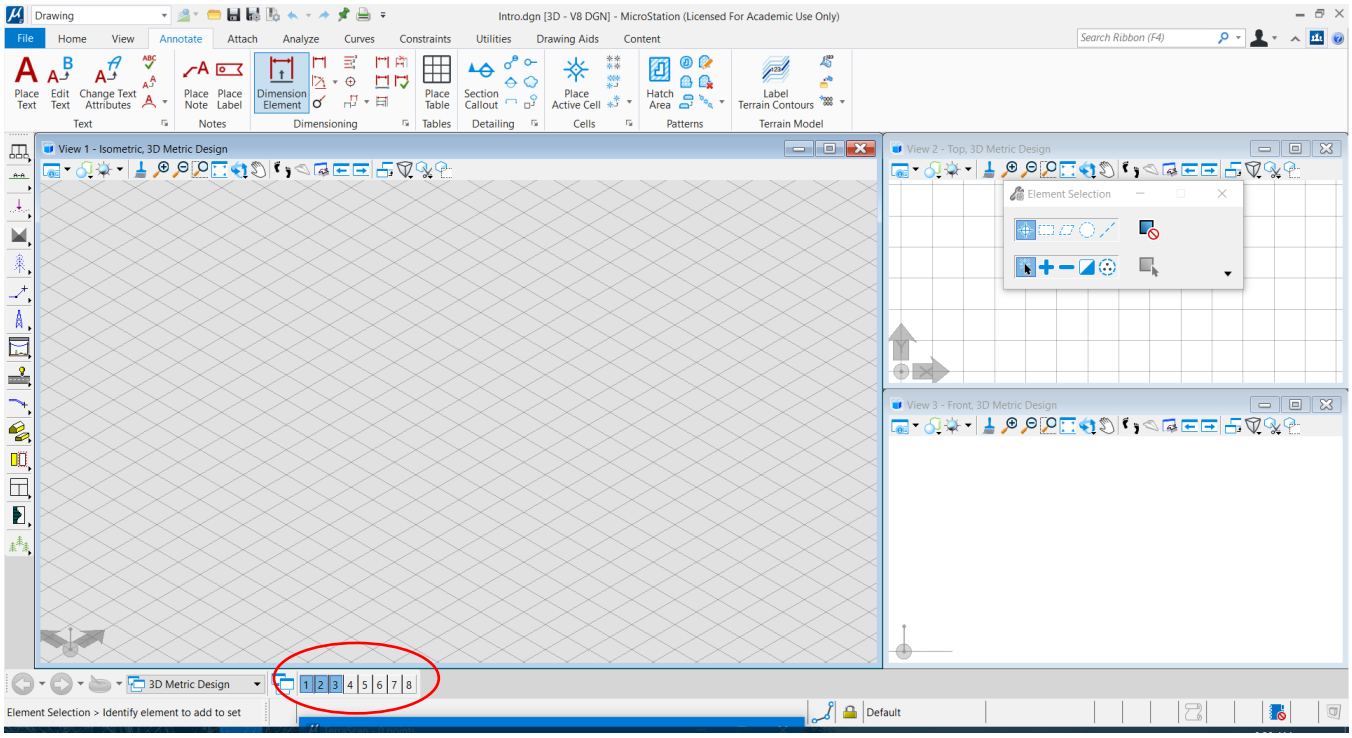


Note however, that the new DGN file is created based upon a "seed" file. Ensure that the "`3D Metric Design.dgn`" file is selected and if not, it can be found under the `C:\Program Files\Bentley\MicroStation CONNECT Edition\MicroStation\Default\Seed\3D Metric Design.dgn` folder. Selecting a 3D seed is **critical** when working with LiDAR data!

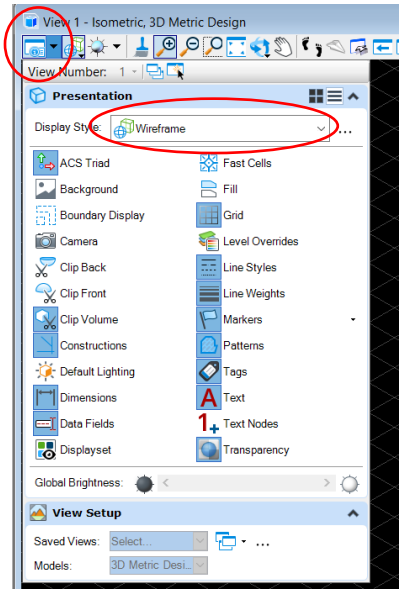
Microstation Views

Perspectives

Once the DGN is created you will be presented with three windows (views) that can be used to display data from various perspectives.



The largest View shows data from a "planimetric" angle, that is from straight above (top down); while the others may be set to other perspectives. Multiple windows do not have to be open as we work through a project, so it is suggested that all views other than the main view are closed as they can be opened at will by enabling a view via the buttons circled above.



Note that the background in the screenshot above is white. This is not desirable when working with a point cloud. A black background will enable a user to more effectively see the points.

To set the background as black, select the *View Attributes* button, which is located at the top left corner of each view. From here, change the *Display Style* to *Wireframe*. You can control many other view options from the View Attributes window as well.

Navigation

Along the top of each view is a variety of tools that can be used to navigate through your data. Below is a brief description of the more commonly used tools:

Update View:  Refreshes the display; all levels (i.e. layers) are redrawn.

Zoom-In:  Zooms into an area specified by a fixed sized window. Right-clicking ends.

Zoom-Out:  Zooms out a fixed amount.

Window Area:  Allows you to click & drag to specify an area to zoom into.

Fit: View:  Shows all the data in the DGN file in the zoom (i.e. world view).

View Rotation:  Allows you to rotate 3D data.

Pan View:  Click location to shift, then click location to shift to.

View Previous:  Returns to the previous view extent.

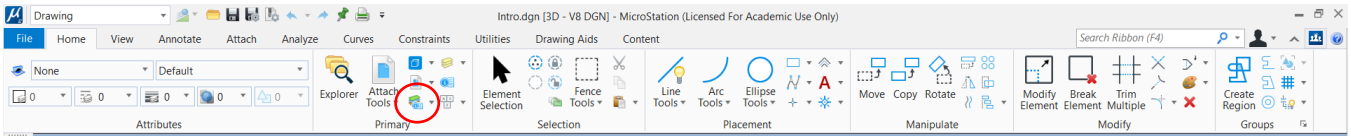
View Next:  Re-does the most recent view change.

Managing Levels

In Microstation data is displayed in "*levels*". You are able to place your various types of data (e.g. area of interest, blocks, flightlines, etc.) into separate levels in order to help organize your DGN. Note: LiDAR data is not displayed on *Levels.* Only data that has been drawn into the DGN are on levels.

Level Manager

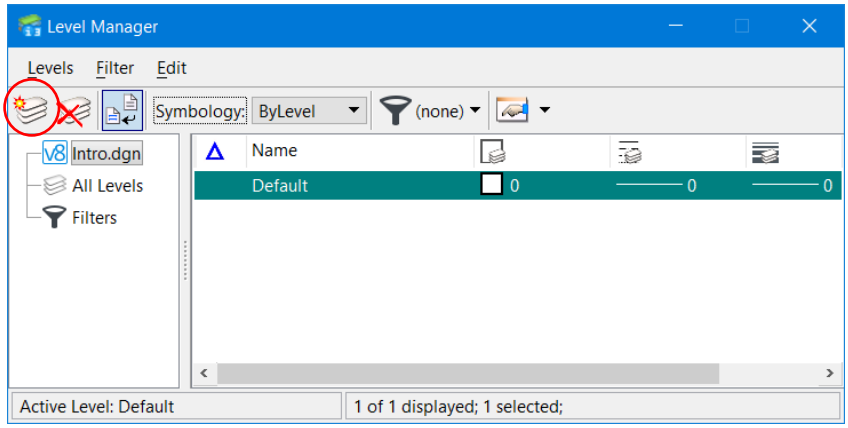
Levels can be created, viewed and deleted through the **Level Manager**, which may be accessed on the *Home* ribbon in the *Primary* section.



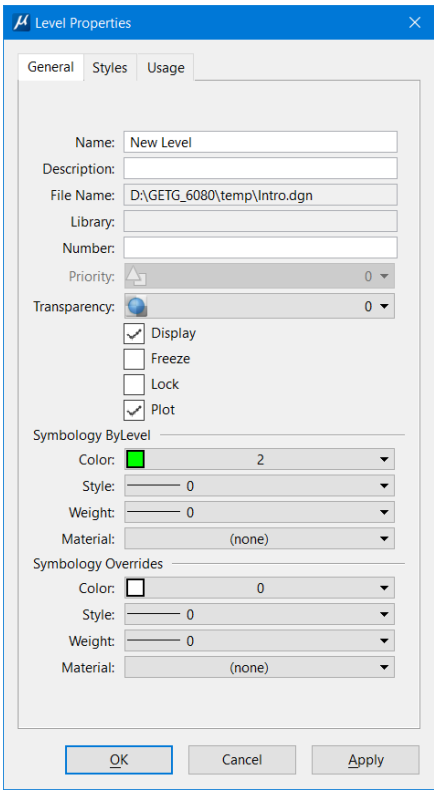
A new level can be created by selecting *New* from the *Levels* menu or by clicking the *New Level* button (circled right).

Similarly, a level may be deleted by selecting the *Delete Level* button.

Note however that only levels that are unused (i.e. contain no elements) can be deleted.

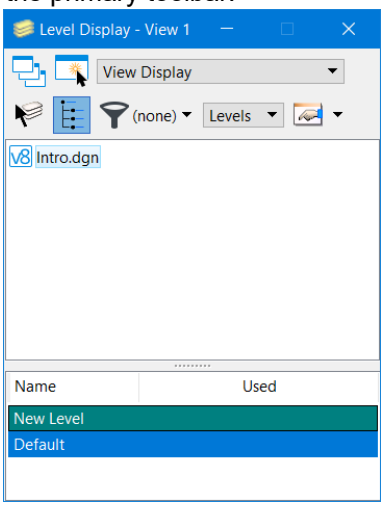


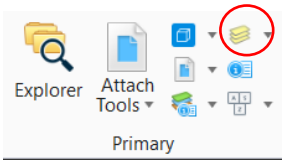
Other aspects such as the color, number (display order) and line style and weights can be changed by right-clicking on the level of interest and selecting *Properties* (right).



Level Display

While the *Level Manager* allows you to create and change levels, the **Level Display** window (left below) is used to turn on/off particular levels. This can be accessed through the *Level Display* button (circled below right) in the primary toolbar.





The bottom half of the window displays the levels in your current DGN. Levels highlighted in blue are being displayed while levels being hidden are not highlighted. At any given time, one particular level is active (i.e. for editing) and it is highlighted in green.

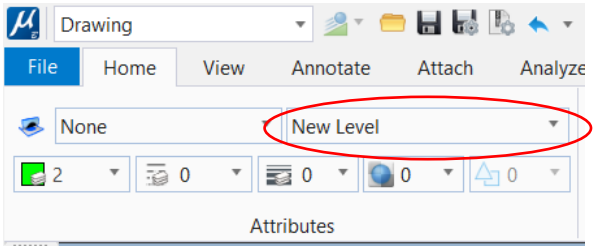
Clicking on a level will change its visible status; however it may be necessary to manually refresh the views to show the change (e.g. using the *Update View* button along the top of the *View*'s window).

You may also change the active level by double-clicking on another level. This can also be changed through the *Set Active Level* drop down discussed in the next section.

Managing Elements

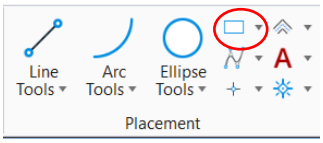
Active Level

Whenever a new element (i.e. somewhat like a feature in GIS) is created it is placed within the currently active level. The active level may be changed within the *Level Display* window described above, but can be more easily accessed through the *Set Active Level* dropdown (circled right).



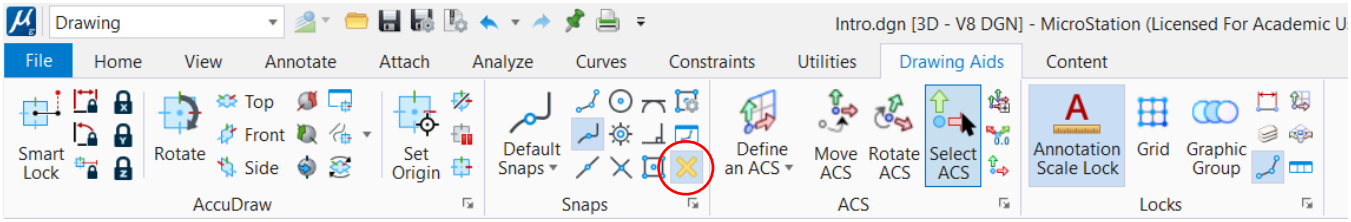
Creating Elements

Microstation allows you to manually create elements such as points, polygons, boxes, lines, arcs, circles, etc. through its various "*Placement*" tools.



Polygons can be created to delineate areas of interest (e.g. to restrict processing, locations to observe, etc.) using the *Place Shape* tool (circled above).

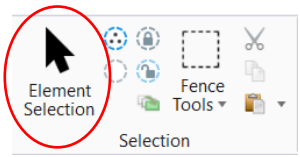
Before drawing, the user should activate *AccuSnap* to ensure that drawings will snap to existing vertices. *AccuSnap* can be found in the *Snaps* section of the *Drawing Aids* ribbon (circled below).



After activating the *Place Shape* tool, and enabling *AccuSnap*, clicking within a *View* will create a new vertex. Clicking back on the first vertex (it will automatically snap when you're close enough) closes the polygon and adds it to the active level. Right-clicking at any time will cancel the polygon and nothing will be added to the DGN.

Note: in order to best maintain **compatibility with ArcGIS** (e.g. when importing your DGN data) be sure to digitize your vertices in *clockwise* order (otherwise you may need to use the ArcGIS *Repair Geometry* tool).

Selecting Elements



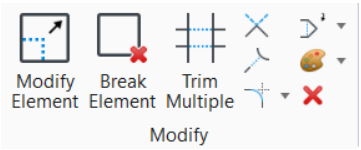
The *Element Selection* tool (circled right) is used to select one or more elements. Many Microstation (and TerraScan) operations require certain elements to be selected (e.g. one LiDAR block for processing).

Clicking on an element will select it (it should be obvious when the element is selected) and you may include additional elements in that selection by using the Ctrl key in conjunction with clicking on the new ones. A group of neighbouring elements can be selected by clicking and dragging a selecting box around all of them (much like selecting vector features within ArcMap).

Deleting Elements

Selected elements can be removed from the DGN using *Delete Element*  (found under the *Modify* section of the *Home* ribbon) or by pressing the Del key

Modifying Elements



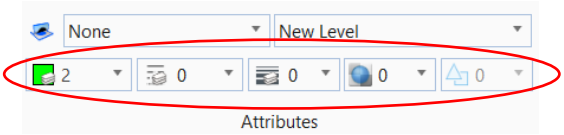
Other aspects of an element can be edited via various tools in the *Modify* section (right) of the *Home* ribbon. The vertices of an element (e.g. polygon) can be manipulated through the *Modify* toolbar. There are several tools that allow you to move, insert & delete vertices. Once a tool is activated, move to the vertex to change (or where to insert one) and left-click to begin the process. Right-clicking cancels the operation without making changes.

Changing Levels

Elements can be moved to another level by selecting them (see *Selecting Elements* above) and then changing the active level through the *Set Active Level* dropdown (see *Active Level* above).

Element Symbology

Beneath the *Set Active Level* dropdown are three additional dropdowns that allow you to change the color, style or thickness of any selected element(s).



Reference Data

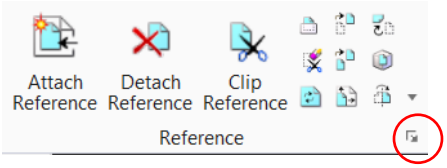
Generally most of the data you work with will be stored within your DGN; however Microstation does allow you to load and display external data for reference.

For example, in order to help you create LiDAR blocks (described later) you may wish to import a study area polygon from an external source (e.g. polygon vectors) into your DGN. While Microstation can read a variety of

CAD file formats, it has limited support for common GIS formats (there are extensions that enhance this). Fortunately ArcGIS (and many other packages) can export data into Microstation DGN files.

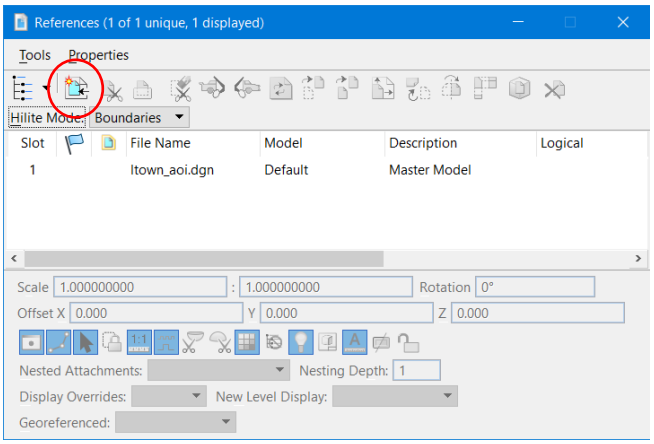
Creating a Reference Dataset

The *Reference* toolbar can be found in the *Attach* ribbon (right). To access the *References* window the user must click the *References* button (circled right).

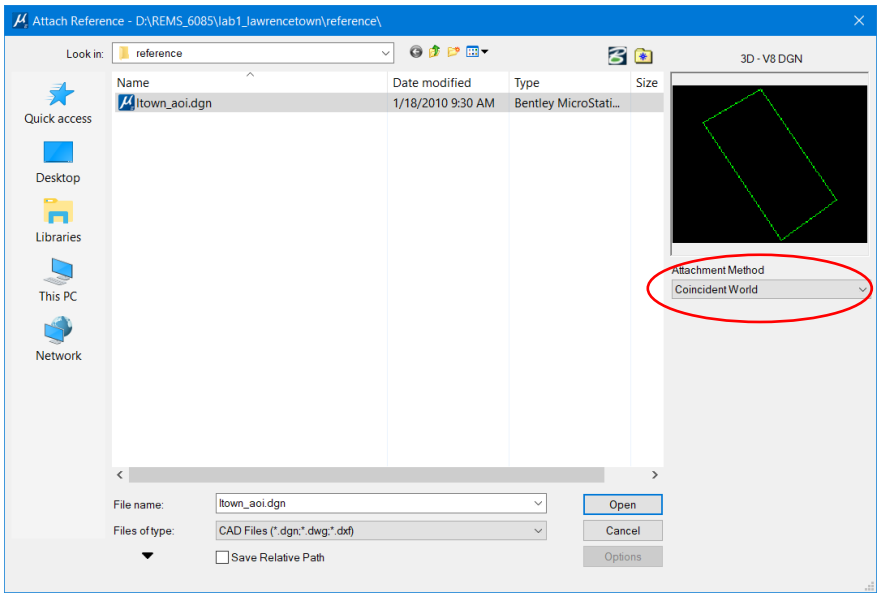


The *References* window (right) is used to add, remove and manage reference datasets (external files sharing the same spatial extents as your DGN).

Add a new reference dataset by clicking on the *Attach Reference* button (circled right) to open a new file selection window (below) and locating the DGN file containing the data you wish to use (`ltown_aoi.dgn` in this case).



If the reference data uses the same coordinate system as you DGN (typical situation) then set the *Attachment Method* to *Coincident World (circled right)* to tell Microstation to merge the two datasets.



After selecting the file to use as a reference dataset, the file will be included in the *References* list.

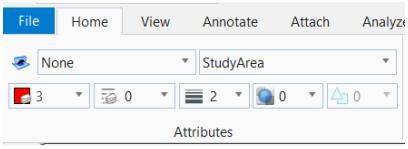
References may be displayed and hidden much like other levels, but may also be imported directly into the DGN (next section).

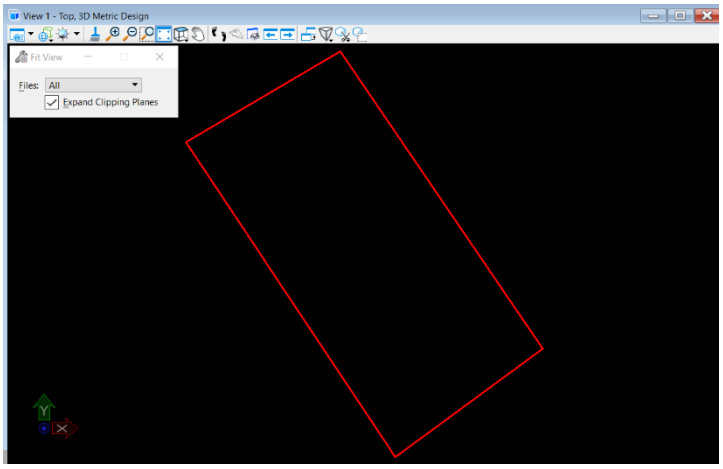
Importing a Reference Dataset

Once added as a reference, you may import the elements into your DGN. Within the *Reference* window, right-click on the reference of interest (e.g. `ltown_aoi.dgn` above) and select *Merge Into Master* (found under the *Tools* menu in the *References* window).

Selecting this function will not immediately perform the task, but rather requires you to select a particular view to operate within (a common technique used for many Microstation & TerraScan tools). Immediately after selecting *Merge Into Master*, click somewhere within *View 1* (your primary, planimetric view) to continue the merging.

The imported reference dataset will be placed in the `Default` level. You may wish to reorganize the data by selecting the newly imported elements and choosing a new level (e.g. `Study Area` right) in the *Set Active Level* drop down. This will transfer the selected elements to the specified level.





Once completed, the selected reference dataset is then removed from the *References* window (the data is now part of your DGN instead).

Sometimes you will not immediately see the results of the merge as the view may not automatically update to include its location. Clicking on the *Fit View* button  at the top of the view will show the newly merged data (left).

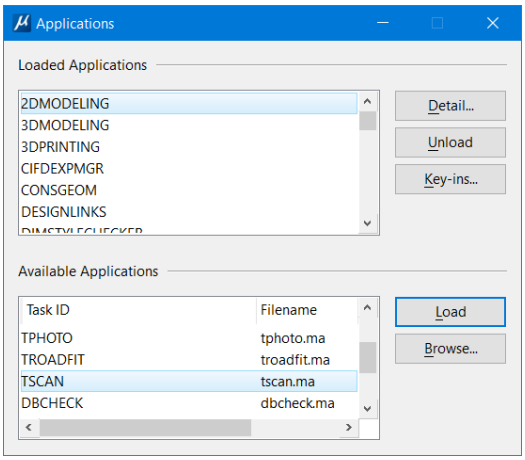
Once the reference data has been loaded, you may wish to use the element editing tools to change the default symbology (e.g. make the AOI polygon red), etc.

TerraScan & Microstation

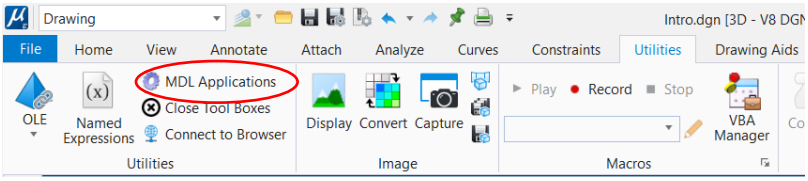
TerraScan has a strong relationship with Microstation in that many of TerraScan's operations make use of Microstation's built-in functionality, yet it provides a variety of its own tools specific for LiDAR processing. The following sections of this guide will introduce you to some of these tools and how the two software packages interact with one-another.

Loading TerraScan

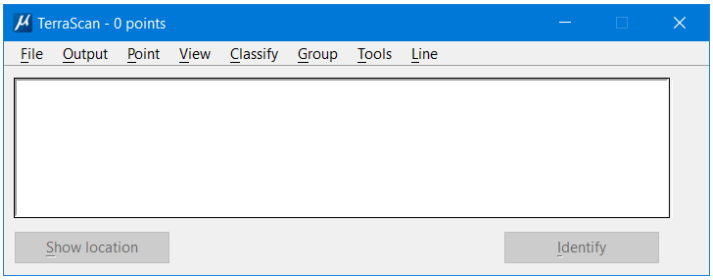
TerraScan is an extension to Microstation that makes use of its Microstation Development Library (MDL). As such, you must load TerraScan before using any of its functionality.



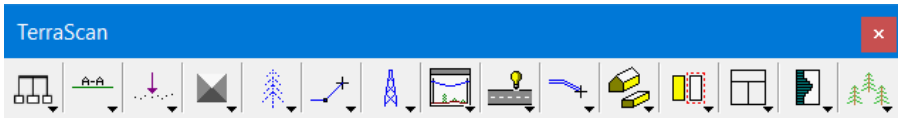
To open the *MDL* window (left), select the *MDL Applications* button (below) from the *Utilities* toolbox that can be found in the *Utilities* ribbon. From here locate the `TSCAN` application (highlighted left). Clicking Load will initialize the TerraScan module. NOTE: a license must be borrowed in order for TSCAN to successfully load.



When TerraScan is successfully loaded, you should see the main window (below)



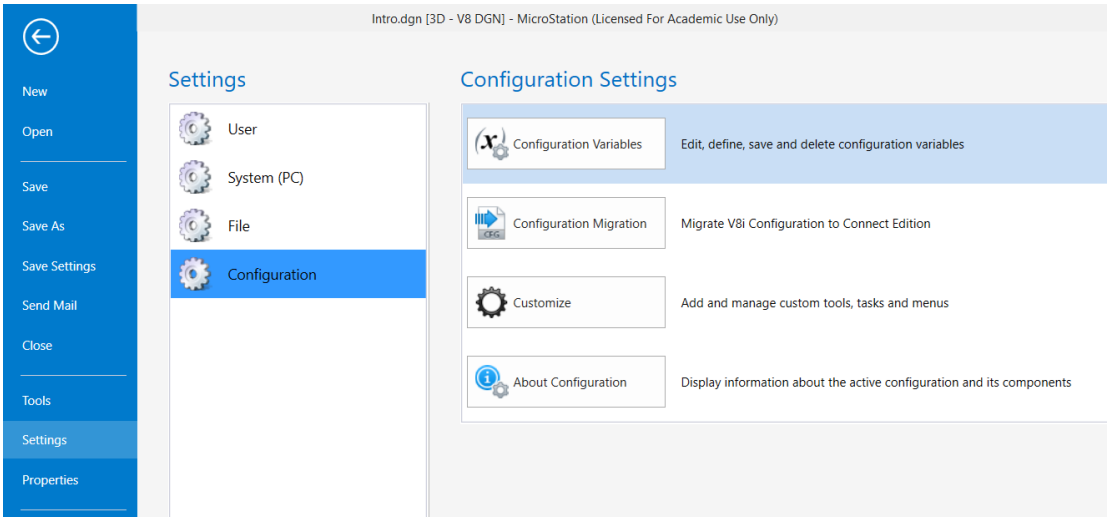
and the TerraScan toolbar (right).



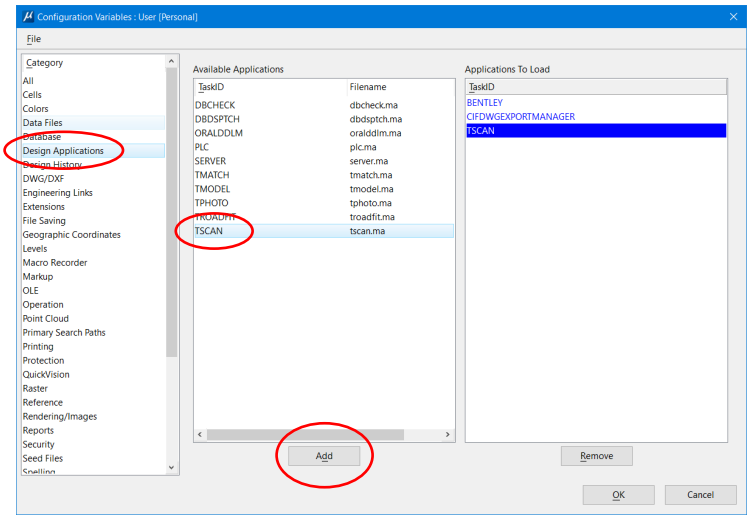
Automatic Loading

Rather than manually loading TerraScan each time you start Microstation, you can set it to be automatically loaded though the configuration settings.

Select *File* from the Microstation ribbon to access the *Settings*. From the *Settings*, select *Configuration* and *Configuration Variables*.



From *Configuration Variables* select Design Applications. This will then populate the middle column with available applications that can be loaded into Microstation. From the *Available Applications* column select *TSCAN,* then click the *Add* button to add TSCAN to the *Applications to Load* column. Click OK to finish.



When Microstation asks you if you want to save changes to your configuration file (`.ucf`) select *Yes*.

The next time you open Microstation TerraSolid should load automatically unless...

Since this file (configuration `*.ucf`) resides on the C drive, some software installations (e.g. computer labs) may not let you keep the changes.

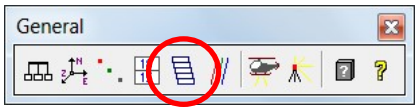
In this case, you will have to use the manual loading technique.

TerraScan Projects

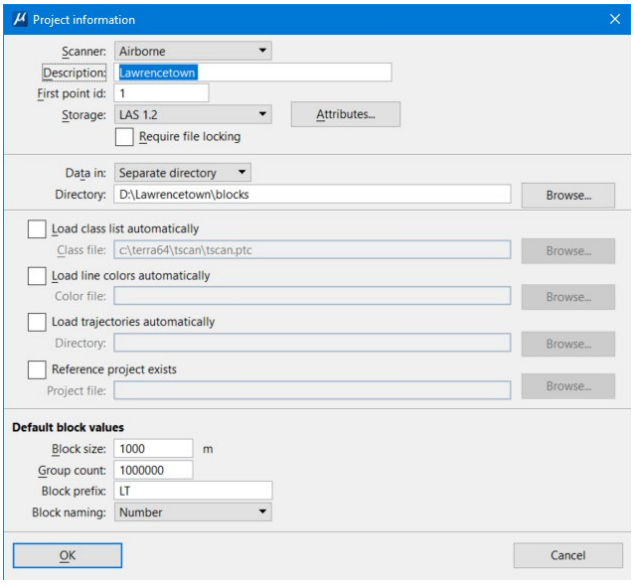
TerraScan allows you to create projects to help keep track of the various files (e.g. block files, to be discussed later) and processing stages often associated with LiDAR surveys.

Creating a Project

Locate the *Define project* tool within the TerraScan toolbar (circled right) and open the **Define Project** window. Create a new project using [*File / New*] and the **Project Information** window (right below) will open.



Specify a description for the project (e.g. `Lawrencetown`), the format the LiDAR data is to be stored in (e.g. LAS 1.2), the block size (discussed later) for the data blocks (e.g. 1000 m) and the prefix for block filenames (e.g. `LT` for Lawrencetown).



Typically all data for a TerraScan project is stored within a parent folder that contains various sub-folders for specific types of data. For now, you should state that the block files are to be stored and found within the `blocks` folder. Other folders will be discussed in upcoming labs.

Once you close this window, the project will be created **but not saved**. Be sure to immediately save your project file using *[File / Save project as]* within the TerraScan *Project* window.

Creating Blocks (method 1 of 2)

As discussed in class, TerraScan organizes LiDAR data spatially into tiles (or blocks) in order to reduce memory usage and processing times. TerraScan projects (see previous section) then allow these blocks to be viewed, managed and processed individually or as a group.

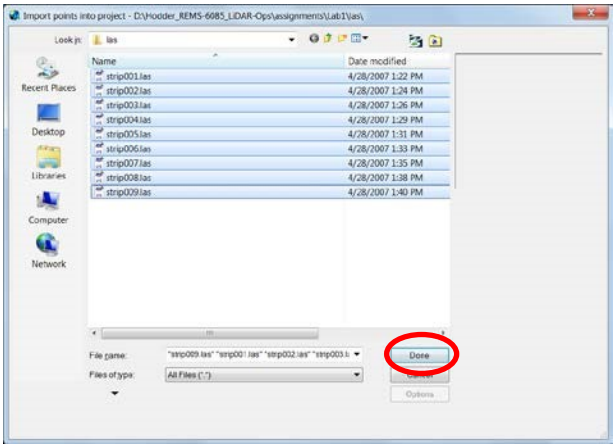
There are two basic approaches to blocks: overlapping and non-overlapping. Non-overlapping blocks are easiest to create and more commonly used in industry. TerraScan offers tools to largely automate the process if there is no pre-existing project boundary file. These steps are described in the following sections.

Loading Points into Project

After creating the TerraScan project (see *TerraScan Projects* above) simply import all of the LiDAR points and TerraScan will automatically create blocks according to the *Block size* specified in the project settings (e.g. 1000m x 1000m). The blocks will be aligned to an "even" block size (e.g. Easting of 334000; not 334123.4) and will span the data extents to include all of the input points (e.g. every point will load into a block).

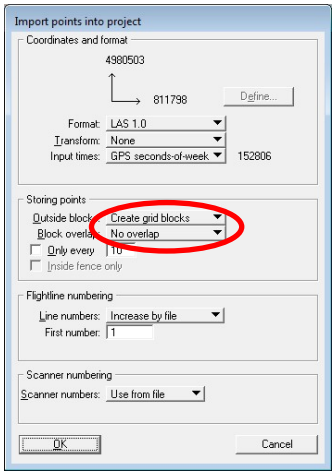
During the import, a new file (e.g. LAS) will be created for each block in the project (with the specified *Block prefix*, e.g. `LT`) and stored under the folder you specified while creating the TerraScan project (e.g. `blocks` folder).

Import the LiDAR points by selecting *Import points into project* under the *File* menu of the *Project* window. This will open a file selection dialog (right) that allows you to select one or more input files (e.g. LAS). Navigate to the folder that contains your LiDAR points (`.las` in this case), select all of the files and press the *Done* button.

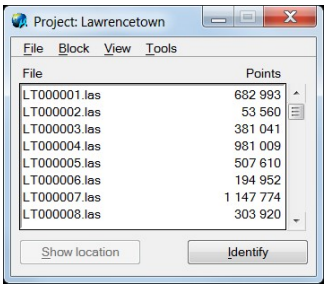


Once you've added the files to import, pressing *Done* will close the import dialog and present you with additional importing options (below right).

To tell TerraScan to automatically create blocks covering the extents of the selected LAS files select the *Create grid blocks* option under *Outside blocks* and the *No overlap* option for *Block overlap* (circled right).



We'll be discussing trajectories (i.e. flightline identification) in an upcoming lab; for now though select the *Increase by file* option to create a pseudo-flightline numbering system (since each input LAS contains points for just one flightline).



Similarly, projection definitions and transformations within TerraScan will also be discussed in upcoming labs, so for now leave the *Transform* option as *None*.

Once the importing process has completed, the *Project* window updates to show the number of points that were stored into each block (left) and a summary report is generated (save this as a text file).

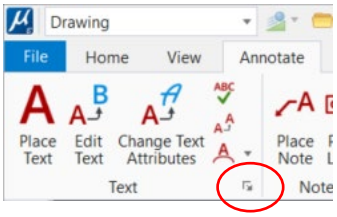
The disadvantage of this method is that the tiles will not be logically numbered. The project tiles should be numbered left to right from top to bottom. Using the LiDAR points to create the tiles causes the numbering to follow the path of the aircraft, which will usually create a less than ideal numbering system. There are methods to reorganize the numbering of the tiles, which may be discussed later.

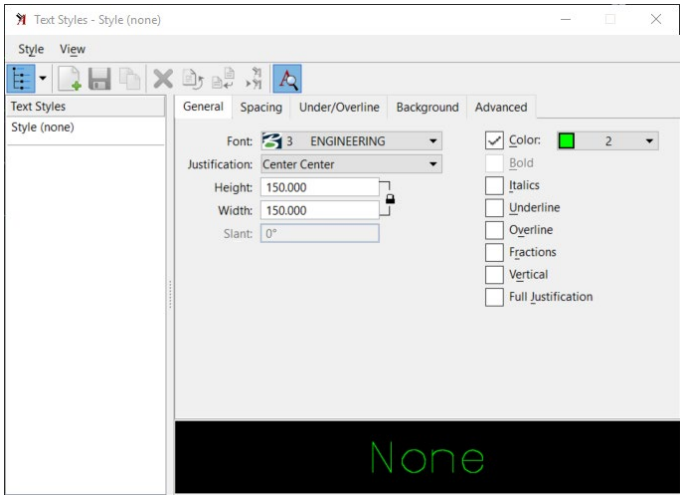
Adding Blocks to Microstation

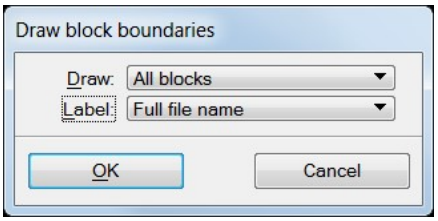
Once the LAS files have been organized into blocks, the TerraScan project now contains information about each block (spatial extents, filename, etc.). Although this is enough for TerraScan to perform automated processing (discussed further in *TerraScan Guide 2*), it is important to add the project blocks as elements to your Microstation DGN file for display & use during processing.

Begin by creating a new level called `Blocks` within the *Level Manager*. Once created, make it the active level by selecting it within the *Set Active Level* dropdown (see *Managing Levels* earlier). Then choose the symbology (e.g. green lines) you wish to apply to the block elements (see *Managing Elements*).

Since labels for each block will be added to the DGN, you should set the text style to use for the labels by opening the **Text Styles** dialog (right) which can be found in the *Text* pane under the *Annotate* tab (circled below). The `ENGINEERING` font style (#3) and size of 150 to 200 is recommended.





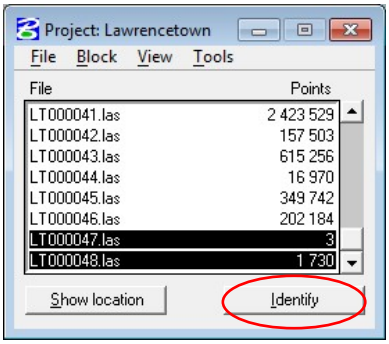


Finally, use the *Draw boundaries* tool (left) under the *Block* menu of the TerraScan *Project* window to "draw" the block extents into your DGN.

You can choose to add all the blocks in the project (typical) or just those you've selected in the *Project* window. You can choose to label the blocks in the DGN according to their full filename (e.g. `LT00034`), block number (e.g. `00034`) or unique end of name (e.g. `34`).

Removing Unwanted Blocks

Blocks in the project with very small numbers of points (e.g. approx< 1000) may cause issues during subsequent processing (e.g. ground classification) and may need to be removed from the project. Similarly, you may wish to remove any blocks "outside" your main areas of interest.



These can be removed from the project by selecting the block(s) in the *Project* window (right) and using the *Delete definition* tool under the *Block* menu.

You should also remove the element and label (see *Managing Elements* earlier in this guide) related to that block as well as the block file (e.g. LAS) that was produced for it (within the `blocks` folder).

You can also remove unwanted blocks by clicking the *Identify* button (circled above), and while holding down the *ctrl* key, click inside of the blocks you would like to remove. Once the undesired blocks are all selected (be sure to double check that blocks that should be included in the project will not be removed), they can be removed from the project by selecting *Block* | *Delete definition* from the project window. Save your project so the latest block definitions are saved.

Creating Blocks (method 2 of 2)

While TerraScan can create blocks automatically (see previous section) if you wish to create blocks based on a pre-existing project boundary file, you must manually create the blocks within the Microstation environment.

This method involves determining the extent of the data to be imported (i.e. project boundary file), then building a single block element. Once a single block has been created, it can then be used as a template to build a

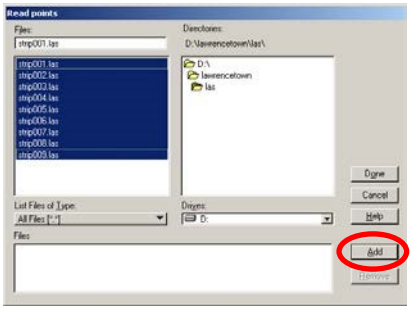
matrix (array) of additional block elements. After the elements for each block have been created, they can then be imported into your TerraScan project and used to import the LiDAR points.

Creating Block Elements

Level Preparation

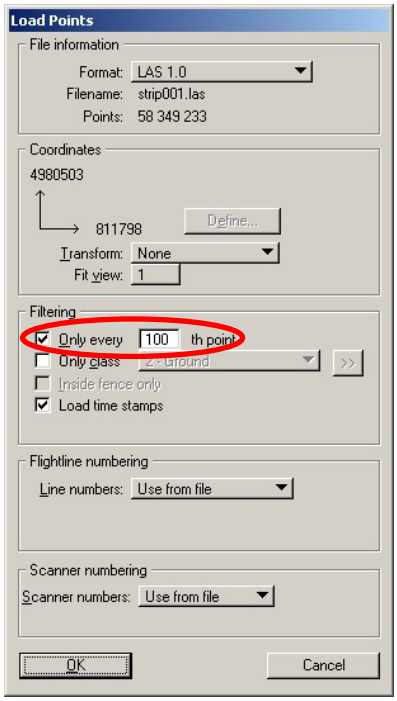
Begin by creating a new level called `Blocks` within the *Level Manager*. Once created, make it the active level by selecting it within the *Set Active Level* dropdown. Finally, choose the symbology you wish to apply to the block elements (e.g. green lines).

Using LiDAR Points



If a study area polygon is not available (or only a rough one), then you may wish to query the LiDAR points directly.

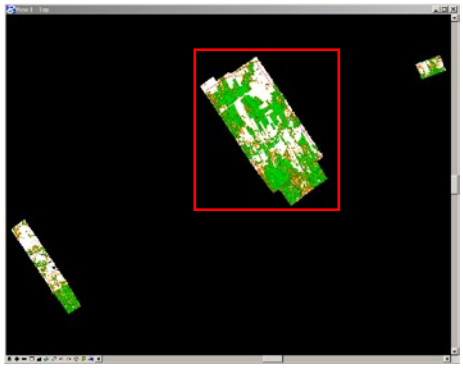
Unfortunately due to the sheer number of points often associated with LiDAR surveys you cannot simply load them all into memory for display. Instead only a sampling of points will be loaded.



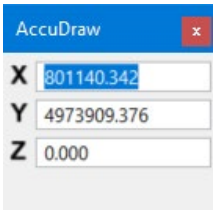
Select *Read points* from the *File* menu in the main TerraScan window, or select the *Load Airborne Points* button  from the *Main* TerraScan tool bar, navigate to where your LAS files reside (e.g. `D:\lawrencetown\las`), select **all** of the LAS files and click on *Add* (above). Clicking on *Done* then opens the **Load points** window (right).

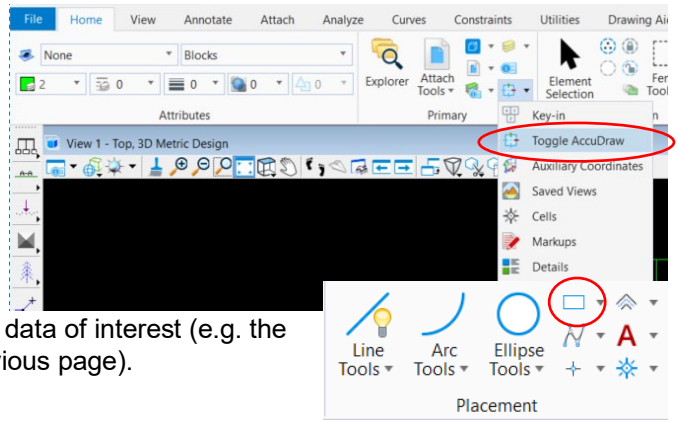
While this window offers a variety of processing options (some will be discussed in upcoming labs), for now click on the *Only every* option (circled right) and tell TerraScan to load only every 100th point.

Once the data has finished loading (i.e. every 100th point), it will be displayed within *View 1*. However, the extents of the *View* may not match your loaded data. Clicking on the *Fit* button in the TerraScan *Display Mode* window will zoom to the loaded points (right). This sampling of data can act as boundary file if there isn't one available.



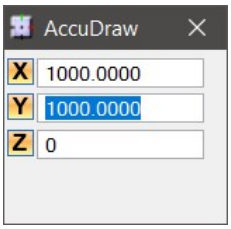
Next, open the *AccuDraw* tool by clicking on its icon (circled right) in the *Primary* toolbar under the *Home* tab. When you move the cursor into *View 1* you will see it display the corresponding Easting & Northing displayed within the *AccuDraw* text fields (left).

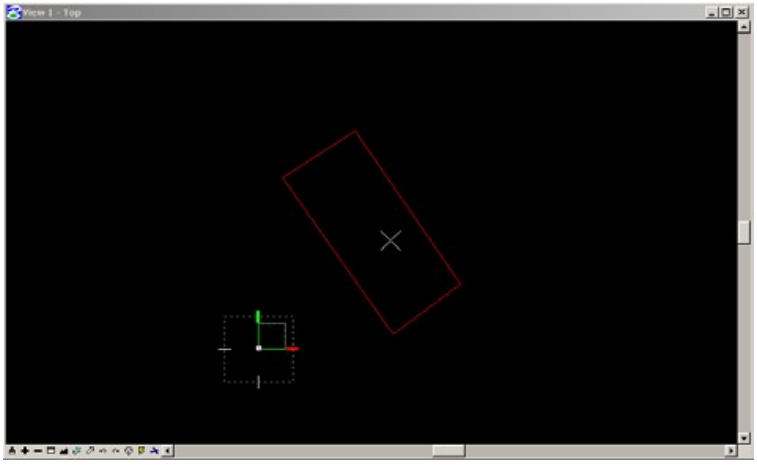




Select the *Place Block* tool (circled right) and move the cursor to the lower left corner of your data of interest (e.g. the "main" portion of the data within the rectangle on the previous page).

After you click to start drawing your block, enter 1000 into the X and Y *AccuDraw* fields to ensure that the drawn block will be 1000m x 1000m (typical size for a LiDAR block). Then click anywhere in the view to finish drawing the block. You should see something similar to the screenshot below.

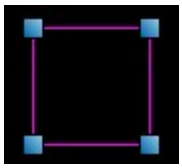


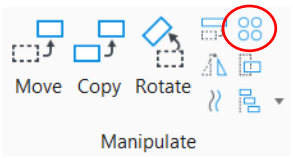


The 1 km x 1 km LiDAR block makes the blocks large enough to keep the number of blocks to manage small, but small enough to be able to load their contained points in memory for processing. For high point density surveys, the block size may need to be smaller.

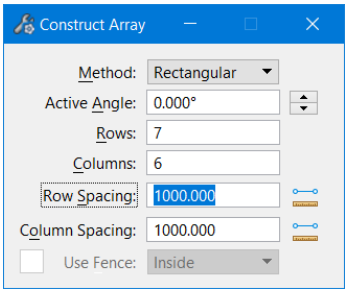
Creating the Remaining Blocks

Once a corner block has been established, it can then be used to generate an array of similarly sized block elements. Begin by using the *Element Selection* tool to select your newly created block (right).



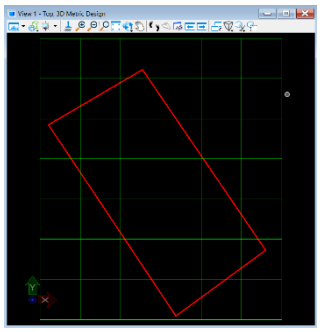


Next locate and select the *Construct Array* tool (circled left) to open its related window (right below).



Within the **Construct Array** window you can specify the number of rows and columns of the array to create.

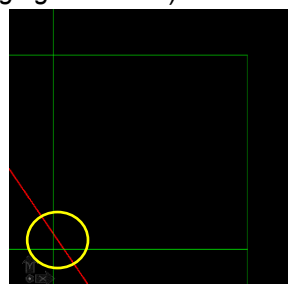
Once you have supplied the various parameters, click the lower left corner or the initial block to designate it as the reference point for the array. Then click anywhere inside of the view to accept, and Microstation will draw an array of 1km x 1km tiles that should cover the area of interest (right).



Removing Unneeded Blocks

When the block array is viewed along with the study area polygon (bottom of previous page) it can be seen that several blocks will not actually contain any LiDAR points.

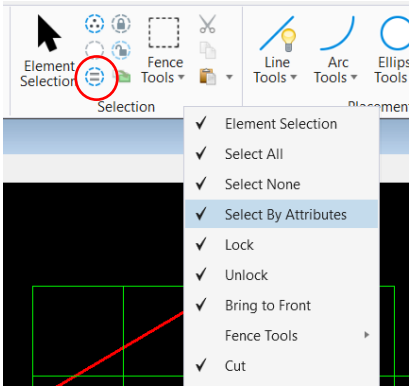
Select and delete any un-needed blocks (right) using the element editing tools (see *Managing Elements*). It is also important that any blocks that are very small on the project boundary (circled right) are adjusted for when possible. Very small tiles will not classify correctly.



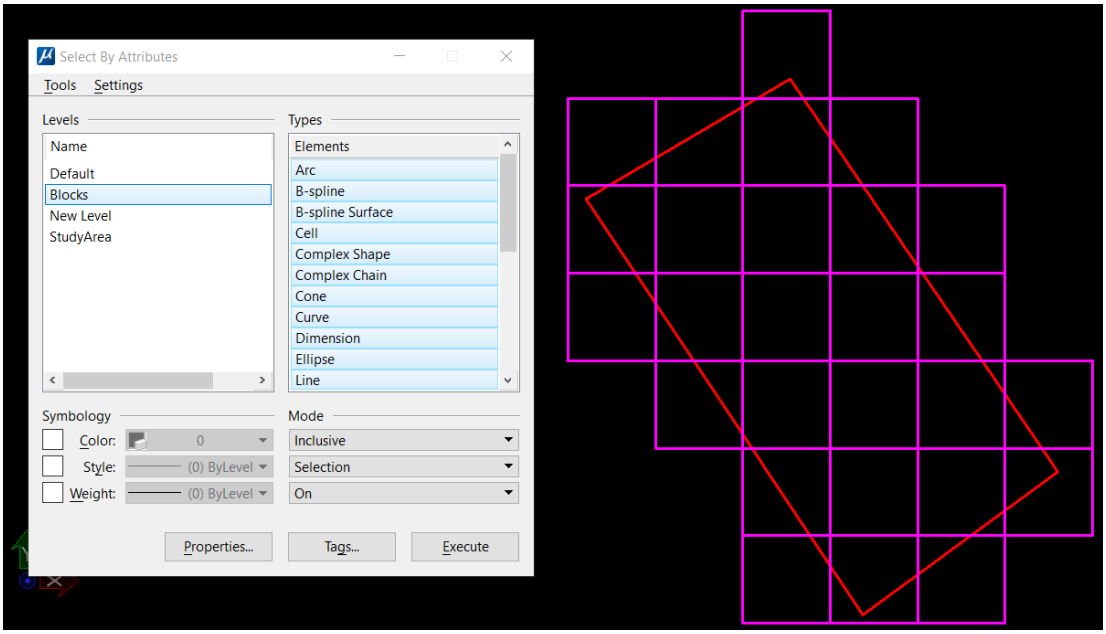
To adjust for this, the user can select all of the blocks (excluding the project boundary or AOI) and manually shift them until any small blocks are eliminated.

There are a couple of methods that can be used to select the blocks. The first method being manually select the blocks using the *Element Selection* tool. The second (and more efficient) method being the *Select by Attributes* tool.

The *Select by Attributes* tool must be activated via the *Selection* pane by right-clicking on the pane and toggling on the *Select by Attributes* option (right). The tool can then be accessed via the *Select by Attributes* button (circled right).



To use the tool, the user must select the level of interest (*Blocks* in this case) and click *Execute*. This will select all elements on the selected level as seen below.

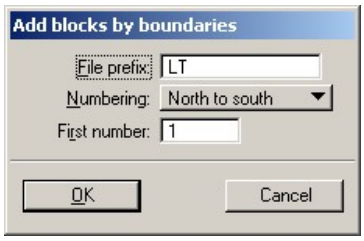


Creating Project Blocks

Once the blocks have been defined and finalized (i.e. all unwanted block definitions have been removed from the project) we must then import the las flightlines into the TerraScan project through the *Project* window. Before we do this we can either make a new directory in which to store our newest project points, or go into the existing `Blocks` directory and delete all existing las files.

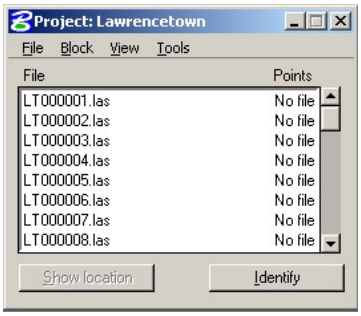
Importing Block Elements

Begin by selecting all of the block elements.



Once the block elements have been selected, use the *Add by boundaries* tool (left) under the *Blocks* menu of the *Project* window to automatically recreate each block based on your new project definition to your TerraScan project.

Before we make out new block boundaries we must delete all existing block boundaries in the project by selecting them all, then choose *Block* | *Delete definition* from the TerraScan *Project* window.



Next provide a *File prefix* (e.g. `LT` for Lawrencetown) and specify a spatial *Numbering* method (`North to south`).

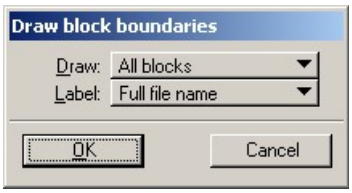
Upon running the *Add by boundaries* tool, TerraScan will then import all of the selected block elements into your project file (above right). Note that although the block extents are now defined, they are not labeled, making it difficult for you to identify and work with the blocks.

Labeling Blocks

Once the block elements have been imported into the project (and automatically given filenames), the DGN should be updated to include those block names.

Set the text style to use for the labels by opening the **Text Styles** dialog (left) through the *Annotate* | *Text Styles* menu. The `ENGINEERING` font style (#3) and size of 150 to 200 is recommended.

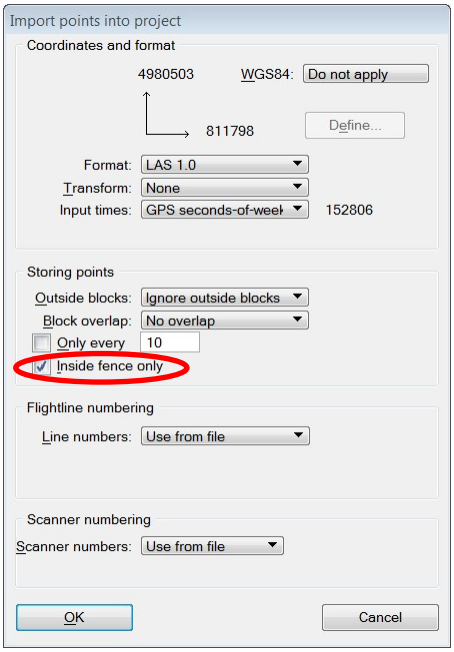
Next, set the active level to where you want the newly created blocks to reside (e.g. `Blocks`).



Finally, delete your existing block elements (trust TerraScan!) and then regenerate them using the *Draw boundaries* tool (right) under the *Block* menu of the TerraScan *Project* window.

Loading Points into Blocks

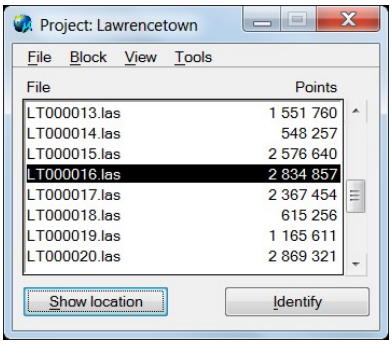
After you have imported the block elements into your TerraScan project, you can now import the LiDAR points into those blocks. This process is similar to loading points as discussed earlier (refer to that previous section for additional details).



Select the *Study Area* element, then use the *Import points into project* tool (under the *File* menu of the *Project* window) to first select which LAS files you wish to import and then provide details on how you wish TerraScan to organize the points (right).

Before importing the points, check the *Inside fence only* box (circled right). This will ensure that we will only import points that fall within the selected *Study Area* boundary.

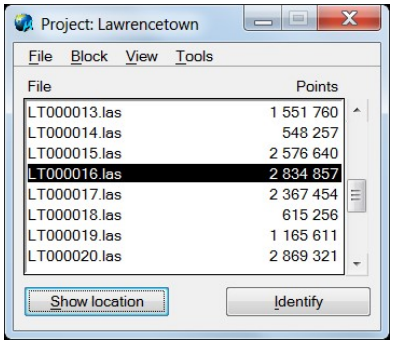
Once the importing process has completed, the *Project* window updates to show the number of points that were stored into each block (below) and a summary report is generated (save this).



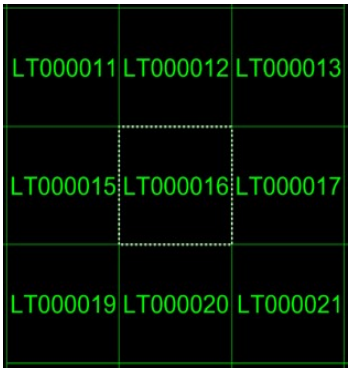
Working with Blocks & Points

Querying Blocks

Although the DGN displays the extents and labels for each block in the TerraScan project, it can be tricky to identify a specific block of interest. Fortunately, TerraScan offers a couple of quick block querying tools.



Show Location



To show the location of one or more blocks, select them within the *Project* window (right above) and then click on the *Show location* button. Move the cursor into the *View* displaying the blocks and the outlines of each selected block will be displayed in a bold white rectangle (right).

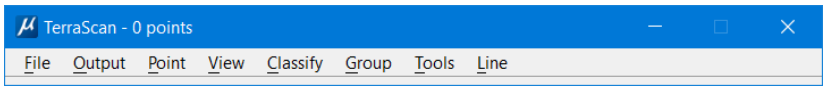
Identify

Similarly, if you click on the *Identify* button in the *Project* window and then click on one of the blocks displayed in a *View* (right) it will select that block within the *Project* window (right above).

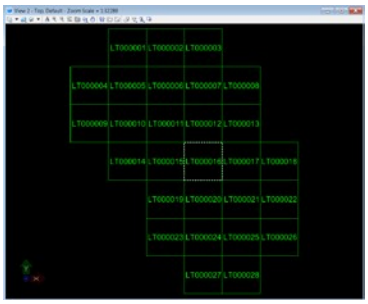
Loading & Displaying Block Points

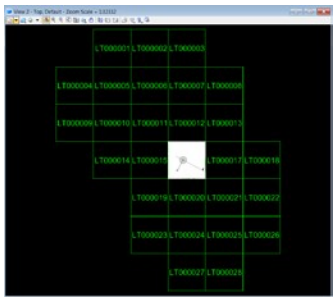
Now that your LiDAR points have been organized into blocks you may now load, display and process the points within these blocks using TerraScan.

The *Open block* option under the *File* menu within the main TerraScan window (right) can be used to load LiDAR points for a specified block.

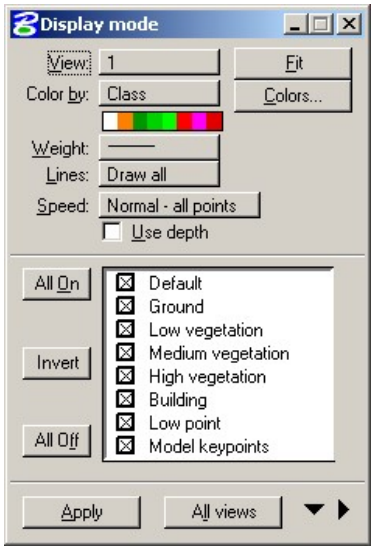


Once *Open block* has been selected, placing the cursor within a block in *a View* (right) will highlight indicating that you wish to load those points (far right). Only a single block of points may be in memory at any time.





Once loaded, you may symbolize the points based upon elevation, intensity, flightline, etc. through the **Display mode** window (under the *View* menu of TerraScan's main window).



The *View* parameter allows you to specify which view you wish to apply your changes to.

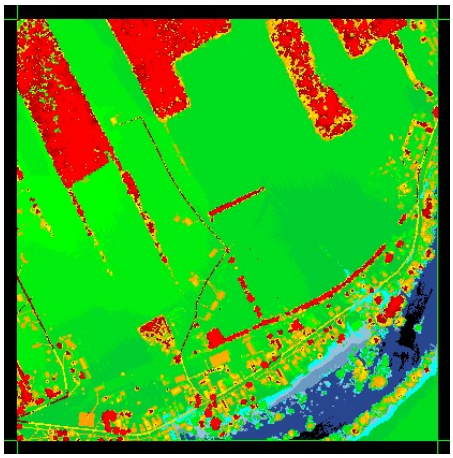
The *Color by* parameter allows you to change how to color the points (e.g. by class, flightline, intensity, etc.).

The *Colors* button opens the **Coloring scheme** window that allows you to specify the range of colors to use. Within this dialog in an *Auto fit* option that will calculate a reasonable range of colors for the data currently in memory.

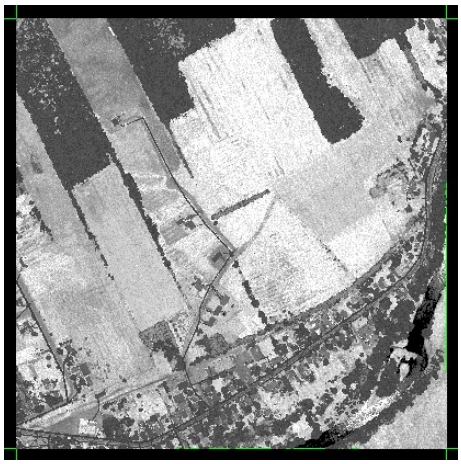
The *Speed* option allows you to display points more quickly; though be careful as not all of your points may be displayed. Set to *Normal* for typical use.

The list of classes (Default, Ground, etc.) allows you to toggle on/off particular classes of points. For example, show only "ground" points in the display.

Finally, the *Fit* button allows you to quickly change the extent of the view to fit the extent of the block currently loaded into memory.



Points by elevation (auto fit)



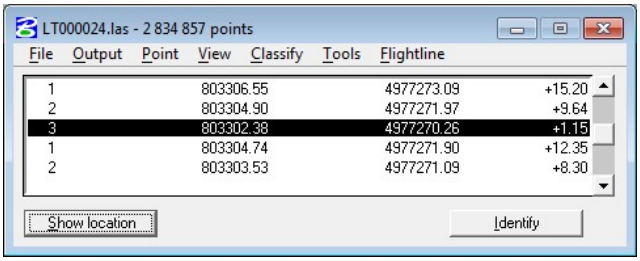
Points by intensity (auto fit)

Querying LiDAR Points

Once points have been loaded you may use the main TerraScan window to view & query specific LiDAR points in the block similar to how you can with blocks (see *Querying Blocks* earlier).

By default the TerraScan window is in “small” mode; that is only the menu items are shown. Selecting *Medium dialog* (or any of the others) under the *View* menu will resize the window to show all of the loaded points.

Each LiDAR point is listed along with various attributes (right); these can be set using the *Fields* option under the *View* menu.



You may locate a particular point of interest (e.g. one with an unusual elevation) by selecting it within the list (above) and pressing *Show location*.

When you move your cursor into the display window (i.e. *View 3*, a profile view - discussed further in *TerraScan Guide 2*) it will display the location with a red “cross-hair” (circled left).

Alternatively you may use the *Identify* tool to select a single point (e.g. one appearing to be mis-classified) and have TerraScan find & select it in the list.