



AUTOMATIC TERRAIN EXTRACTION WITH DENSE POINT MATCHING (EATE) eTRAINING

Introduction

Demonstrates how to automatically extract terrain data using Dense Point Matching (eATE) in IMAGINE Photogrammetry.

Software

- IMAGINE Photogrammetry

Data

- Ireland Quarry

Transcript

0:09

Hello and thank you for watching this Hexagon Geospatial eTraining module Automatic Terrain extraction with Dense Point Matching (eATE).

In this module you'll see the prerequisites for automatic terrain extraction, see the interface for the Dense Point matching algorithm and understand the settings and options to optimize your output DTM accuracy.

0:28

The **Automatic Terrain Extraction** module has 3 algorithms available, **Sparse Matching (ATE)**, **Dense Matching (eATE)**, and **Semi Global Matching (SGM)**.

Before running any of the Terrain extraction algorithms, a block file must be created and triangulation accepted. A block file contains raw imagery and all information needed to run triangulation –including imagery, camera or sensor information, ground control when needed, and elevation data. By running triangulation, a model is created to define the relationship between the images and the ground space. Terrain extraction cannot take place until triangulation is complete.

1:07

You can check your triangulation results by clicking **Block Triangulation** on the **Photogrammetry** tab. The summary opens and you can see the value for the **Total Image Unit Weight RMSE**. This value is an indication of the block accuracy and whenever possible should be kept under a value of 1. With raw imagery, this tells me the accuracy will be within 1 image pixel. In this case the value is .1037 – well under a value of 1.

1. If you haven't already done so, click **Accept** and **Update**.
2. In this example, that's already been done, so click **Close**.
3. To launch the Dense Point Matching algorithm, from the Terrain group on the Photogrammetry tab, select **Generate > Enhanced ATE (eATE)**.

Automatic Terrain Extraction with Dense Point Matching (eATE)

4. And click **Yes** in the Attention box to save any changes to your block.

This is the **eATE Manager**. This interface has all the tools you need to extract your DTM.

2:00

When you start, the images are loaded into the viewer as footprints.

5. You'll start by fitting the images to the window by right clicking in the viewer and selecting **Fit Image to Window**.
6. To see the raster images, you can select all the images in the table and then add a check mark in the **Visible** column for each image that you would like to see. In this case, all of them.
7. Finally, along the top tool bar, you'll turn on the **Display Raster** tool. This will activate all images in the viewer.

2:26

First let's become familiar with this interface. Along the top, there are tools that allow you to modify your display, set preferences, and finally run the DTM extraction. On the left side you'll find the save and open tools. Next are viewing tools – with these tools you can decide to view footprints, rasters, or different types of boundaries. They also include zoom and panning tools. On the right of this tool pallet you find the process engine settings. You'll see more of these in a few minutes.

On the left is the **Project** pane. This pane allows you to define images, AOI regions, and set certain strategy parameters.

3:02

The table at the bottom shows information about each category in the **Project** pane. The information you see here will change based on the category that's selected.

You will become more familiar with these tools as you make your way through the workflow.

The viewer shows the raw images oriented based on the triangulation results. Terrain extraction utilizes the parallax – or relief displacement - between images calculated during triangulation. It's this offset between the images that allows the software to extract the terrain value.

8. You'll start setting your project preferences with the **Project** pane.

3:35

By expanding the Images category, you can see all the images in this project. You can see more information about these images in the table, including file location, height and width, and sensor type.

9. Next you'll click on **AOI**.

3:48

If you want to, you can limit the DTM extent by defining an AOI region. For example, you might only want to extract a DTM for the quarry (quorie) in this imagery. By limiting the DTM extraction area, you can reduce the processing time while extracting the area you are most interested in.

10. You can open the **Region Definition** tools by clicking the icon from your toolbar.
11. Using the tool pallet, you can either define a saved AOI, or use the tools to digitize a new one. In this example, you'll just click **Close**.

4:18

Seed Data provides a starting point for the extraction engine and can increase the accuracy of the output DTM. A surface – or existing DTM, breaklines, or Mass points from a Block file can be used as seed data.

4:31

By default the software will use a global DEM as a seed source. Here you'll also add Mass Points from a block file.

12. Select **Add > Mass points from Block File** and in the File Chooser, you'll select **this DMC 10cm block** and click **OK**. It's now added with the **Global DEM** as a seed source.
13. Below **Seed Source**, select **Strategy** to view the strategy parameters that have been set. You'll take a closer look at this in the **Strategy parameters** dialog box.
14. From the tool pallet, start by opening the **Process Engine Settings**. The settings within this dialog box can have a significant impact on the extraction processing time.

5:11

Stop at Pyramid Level defines which pyramid layers are used in the extraction process. The default defines pyramid layer zero – this tells the software to use all of the reduced resolution datasets available and will ensure your extracted data is as precise as possible. It will also take longer to process. By changing this value to a 1 or 2, your processing time will decrease, but it may also reduce the accuracy of your result. Whenever possible, it's best to leave this set to zero.

5:40

Point Sampling Density can also decrease processing time, but again it can affect the accuracy of the final output. The eATE algorithm is designed to do a pixel by pixel interrogation. Changing this value tells the software to skip pixels and only use every 2nd, 4th, 8th and so-on. This setting can be determined by weighing your accuracy requirements against processing time.

6:04

Pixel Block Size defines the number of pixels considered in the processing at any one time.

If your imagery has fiducial marks, the **Fiducial Mark Offset** setting will trim your imagery so the fiducial marks are not included in the final DTM. If you were to add a value of 10, a 10% border of excluded area would be removed from the output.

6:24

The **Formatting** tab defines settings for the internal processing. If you want to produce an RGB encoded output, be sure there is a checkmark next to the RGB Encode settings.

In the **Radiometry** tab, The Gradient Threshold can be adjusted if your imagery has poor contrast or fidelity. Increasing this value can help the software find match points across the imagery and improve the output results. The imagery in this example has nice contrast so we'll leave the default setting.

6:52

A radiometry layer can be created to visually see how well the radiometry is working when computing the terrain output. This is not necessary, but it can give insight into potential problem areas in your output DTM.

15. Click OK, and next you'll look at the **Output Settings**.
16. From the toolbar, launch the **Output Setting** dialog.

17. From the **General** tab, you can use the file chooser to define where you want the output file to be saved if it's not already defined.

7:20

The **Min Z** and **Max Z** values are threshold values used to constrain the eATE results. All values outside of this range will be discarded. The units are display on the right. These values can be modified, and in most cases the values are derived from the global DEM and any other seed source when being used.

However, if you have is an LSR project, the default values will be -50 and 750 meters. The default values can be changed in the eATE preferences.

By activating the **Bounding Box**, you can clip the output DTM by defining corner coordinates. Or use the button to activate the AOI tools and define an output boundary. This is much like the AOI tool you saw earlier.

8:03

In the **Output Files** tab, you can define which file types to output. When running eATE, you have the option to output three different file types simultaneously - a LAS file or point cloud, a grid file such as a IMG, DTM or DEM, or finally a TIN. All three types can be selected or – as in this case – just one. Today you'll turn off the Raster and TIN options and only output a point cloud.

8:30

The **Split** tab allows you to split the output file into multiple smaller files and provides a couple of different option to do this. You only want a single output file so you'll leave this set to **None**.

The **Thin** tab allows for thinning of the final output data. This can be useful in cases where a system preforms slowly with a very large number of points in the output file. With today's processing power, this is usually not a problem and the defaults are recommended.

8:56

The **Classification** tab defines what will be classified and how the software will identify buildings and vegetation. The default values for building and vegetation parameters usually do a good job at finding these features and are a good place to start. In this case you will use the default values.

18. In **Classification to Keep**, select the classification categories you would like to see in the output. You can select individual values or, as in this case, tell the software to keep all values.

19. You can now click **OK** to the **Output Settings**.

9:27

Next you'll look at **Strategy Parameters**. As a general rule, it's a good idea to leave most of the default values. These values have been determined by engineering to optimize results. However, in some cases, changing the **Interpolation** settings can improve results.

The interpolation method will fill in areas of missing values with one of three options. **Mean** performs an averaging across values, **Region** considers a smaller set of points when performing the interpolation, and **Spike** applies specifically to remove extreme values.

9:59

A smoothing filter is also available to aid in more homogeneous output. The degree of smoothing selected is determined by what features are in your input imagery. Less filtering is recommended to maintain the integrity of your data, especially when the area contains hard edge features such as buildings.

10:16

Finally, if the project imagery has poor contrast, a check mark can be added next to **Low Contrast** to improve the output results.

20. Click close on the **Strategy Manager** and **Yes** to save the changes and overwrite the existing parameters.
21. Finally, click on the icon to produce the processing elements.

10:33

By clicking this icon, a plus sign has been added to the **Processing Elements** in the **Project** pane. By expanding this folder, you can see the configuration files that have been added. Each file defines an element boundary that can be distributed and processed across multiple processors or by using **Condor** to access processing resources on computers on a network. Using distributed processing can greatly reduce project processing time.

10:59

Selecting the CFG file on the project pane highlights the bounding element in the viewer. There are times when the bounding element may be very small, or be very redundant with other bounding elements. In that case, it's ok to delete the configuration file to save processing resources.

22. Now click the icon to open the **Batch Command Editor**.
23. By clicking **Submit**, you can define how and when to process the project.
24. Increase the **Simultaneous Processes** to 3, to use 3 system processors at the same time. In this case, all 3 configuration files will process simultaneously.

11:34

You can also choose to process now, or define the date and time when system resources are more readily available and run the process then.

25. Here, you'll leave **Start Processing Now**, and click **OK**.

Now let's view the outputs.

11:50

Here we see the output LAS file loaded in the viewer. In the upper left, the **Color By** can be changed. As default, the data is displayed as elevation. Using the drop down menu, you can change this to **Classification** to see the classification categories. Finally you'll change the display to **Color By RGB**. This option produces a point cloud display that looks very similar to the original imagery.

26. From the 3D View group on the **Point Cloud** tab, you can select **Show 3D** to open the point cloud in a 3D Viewer. In the 3D viewer, you can tilt and zoom in to get a better view of the features.

12:25

As you continue to zoom in closer to the points, they can become harder to see. In the **Display** group, you can use the **Point Size** tool to increase the size of each point. This helps fill in the space between the points and can improve the image display.

You can also use the options in the 3D View group to change your viewing mode. Selecting the Perspective mode changes the points so they look larger, filling in the space around the points so they appear more filled in. This is similar to increasing the point size.

Automatic Terrain Extraction with Dense Point Matching (eATE)

Triangle mode creates triangle facets between the points, filling in the gaps between the points completely – including the sides of buildings and cliff walls.

13:02

Thank you for watching this eTraining module from Hexagon Geospatial. For more eTraining, visit hexagongeospatial.com/eTraining.