Introduction to Remote Sensing and Photogrammetry

Forestry Applications

Bogdan Strimbu

College of Forestry
Orgon State University

Table of Contents

1 ArcGIS Review

1.1 *Objective*

The first lab has as main objective the refreshment of the main GIS concepts. The focus is placed on data type, spatial manipulation, and data editing.
For those students that have limited knowledge of ArcGIS, you should start by first taking the GIS course online offered free by ESRI: ***Getting Started with GIS***, which is around 3 hours long.

1.2 *Software*

The lab, as almost all the course, uses the GIS software developed by ESRI. There are two possible avenues to complete this lab and the course: ArcMAP or ArcGIS Pro. The lab is developed for ArcMAP, but the concept and interface are almost the same with ArcGIS Pro. For people interested in the free version, the QGIS is an option, but the labs are not designed for that software. Nevertheless, if one or more students want to use QGIS, I would help them.

1.3 *Files Used in This Lab*

The files used in this Lab are located on Canvas under the Week#1 module, the Lab1 link. The files are zipped under the name FE444_Lab01.zip. Inside the zip file there are the following spatial files:

- The shapefile of the stream layer of the HJ Andrews Experimental Forest:
  - HJA_Streams. There are at least 4 files that comprise a shapefile to be used inside ArcGIS, which are distinguished by extensions: .shp, .shx, .dbf, and .prj.
- The shapefile of the boundary of the HJ Andrews Experimental Forest:
  - HJA_Boundary. There are physically 4 files that comprise the shapefile to be used inside ArcGIS, which are distinguished by extension: shp, shx, dbf, and prj.
- The shapefile of the roads from the HJ Andrews Experimental forest area:
  - Roads. There are physically 4 files that comprise the shapefile to be used inside ArcGIS, which are distinguished by extension: shp, shx, dbf, and prj.
- A Digital Terrain Model of the area containing the HJ Andrews Experimental Forest:
  - HJA_DTM.
- A Digital Surface Model of a portion of the HJ Andrews Experimental Forest:
  - HJA_DSM.

1.4 *Geographic Information System*

1.4.1 Spatial data types and formats

Spatial information is usually stored and displayed in two formats: vector and raster. Formally, vector data stores the spatial information as coordinates of particular points, such as vertices of a polygon, and complete the rest of the figure (if needed) by mathematical interpolations. Therefore, vector data are not affected by scale. Vector data can represent points, lines, or polygons. The most common file format used for storing vector data is the shapefile. The shapefile format was developed by ESRI and is a set of three mandatory files: shp, shx, and dbf.

The .shp file is the shape format and stores the feature geometry. The .shx format is the shape index format and stores the positional index of the feature geometry, which allows quick search within the file. The .dbf format is the attribute format, which is a database storing the information within the shapefile. The three files are needed for operability; however, a fourth file is required for meaningful data analysis, namely the .prj file.

The .prj file stores the coordinate system and projection information as an ASCII file. Without the .prj file, we do not know where the information is located and how large/small it is.

The roads layer, stream layer, and the boundary layer are three types of vector data: two are poly-lines (Roads and HJA_Streams), and one is a polygon (HJA_Boundary).

Raster data represents all the spatial information within the extent of the file using a **Cartesian coordinate** system, similar to the vector data, with the difference that the axis units are predefined. The preset of the units discretize the spatial information into cells, which are the natural extension of digitization. Simply stated, a raster is an image, and as such depends on a scale: below the predefined scale there is no information useful within the raster. Depending on the values stored in each cell, rasters have been conventionally classified as discrete rasters, which have a small number of values (such as the one that represents land use), and continuous rasters, which have a multitude of values (such as the one that represents elevation or temperature). Irrespective of the type, the values stored in a raster file is at most the number of cells within the raster; therefore, there is a finite number of values. The most common file format storing raster data is TIFF. However, .jpg or .png are two other formats that can be used. The caveat of using .jpg and .png over .tiff is that they are lossy formats, meaning that information will be lost because of the compression. Furthermore, .jpg and .png have the projection store as a separate file, called world file, whereas tiff sometimes incorporates this information inside the file architecture. The files for this lab called "HJA_DTM" and "HJA_DSM" are continuous rasters stored as tiff files.

Question#1 [10%]: Open ArcGIS and load the five files: HJA_DTM, HJA_DSM, HJA_Streams, HJA_Boundary, Roads. How many features are in the Roads, HJA_Streams, and HJA_Boundary shapefiles?
How many pixels are in the two raster files?

1.4.2 Coordinate systems and datums

In remote sensing, a coordinate system is a narrow application of the mathematical complex coordinates systems to the Earth. Simply stated, a coordinate system enables specification of every location on Earth with a set of three numbers, called coordinates. Intuitively, the coordinates represent a vertical position (distance above a surface or point, if we use the center of the Earth as origin), and two horizontal positions (distances on the surface used as reference). There are two classes of coordinate systems: polar and Cartesian. Irrespective the system, polar or Cartesian, there is an origin and three axes: x, y, and z. For the polar coordinate system, the coordinates of a point on Earth are defined by the origin, also called pole, and three values locating the point: a distance, called the radius, and two angles, usually symbolized by the Greek letters θ and φ. When the angle φ is estimated in respect with the first meridian (aka Greenwich) is called longitude. If the angle θ is computed in respect with the equatorial plane, rather than with the z-axis, basically the complement of θ from Figure 1, then the angle is called latitude.

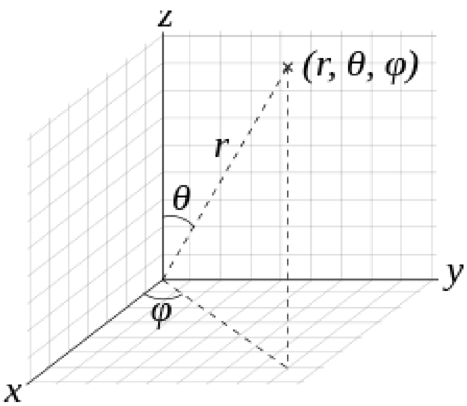


Figure 2. Polar coordinates

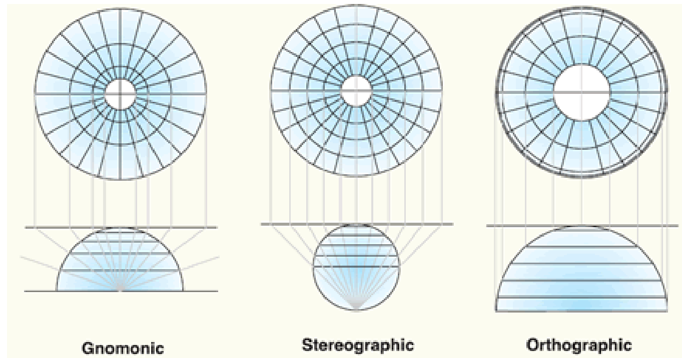


Figure 1. Perspective points for planar projections

Polar coordinate systems are known in GIS or remote sensing under the name of geographic coordinates. The location of the points on curved surfaces, such as Earth, can be exactly computed using polar coordinates, without any loss of information. However, polar coordinates have a major drawback: linear and areal measurements are difficult to execute. These shortcomings of polar coordinates are not existent in the Cartesian coordinates systems. However, conversion between polar and Cartesian systems is not necessarily without sacrifices, which can come under the form of a lack of realistic shape or relationship among land-feature size. We will learn more about coordinate systems later in the class. But for now, what you should know is that conversion from polar to Cartesian, or from a curved surface to a plane is defined by two parameters: the location of the perspective point (also known as viewpoint) and the surface on which the curved surface will be projected on. There are three types of surfaces on which Erath is usually projected on a plane, a cone, or on a cylinder (Figure 2). If the perspective point is placed in the center of the Earth, then the projection is called Gnomonic. When the perspective point is located on the Earth on the point opposing the tangent point between the Earth and the projection surface, the projection is Stereographic. When the perspective point is located at infinity, then the projection is called orthographic. The most popular projection system is UTM, which stands for Universal Transverse Mercator. Lately, another projection system gain traction because of its wide use in GPS technology, namely WGS84.

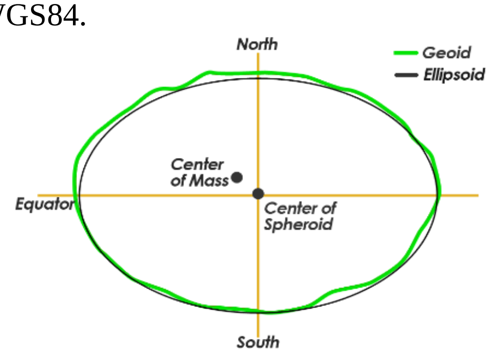


Figure 3. Datum: ellipsoid and geoid.

The first decision is on what projection system to use in locating a point on Earth surface, which in essence will convert the two angles θ and φ into points on a straight line, therefore inside a plane. Once the selection of a projection system is made, next the distance from the pole, meaning from the origin of the coordinate system has to be made. In GIS and remote sensing the distance from the pole of the system is usually estimated not as the distance from the origin but as the distance above a solid of revolution, commonly an ellipsoid (**Error! Reference source not found.**). The object in respect with which the distance of the point is computed from is called datum. Formally, a datum is a mathematical model of the Earth. The datum consists of a set of values that define the shape and size of the ellipsoid.

Question#2 [10%]: What are the coordinate systems of each of the five files (projected or geographic coordinate system)?Convert the coordinate system of all files to WGS 84. Provide a snapshot of the window or code creating the conversion.

1.5 Operations with vector data

Depending on the type of data, many spatial manipulations can be executed. In this section of the lab, we will review some of the basic operations on vector data.

Question#3 [5%]: Clipping – lines with polygons

In many instances, you would like to extract from a larger layer, only the information needed to answer a specific question. The information required to be extracted can be of any type, raster, or vector, which further can be also a point, a line, or a polygon. In this exercise, we will extract the lines following inside a polygon, namely the roads and streams that are within HJA Forest. To do so, you should use the Clip procedure located under the Geoprocessing Menu option. However, before start executing spatial operations, it is good practice to work with files that have the same coordinates system. Therefore, you would use the files converted to WGS84. ***What is the total length of the roads and streams located inside HJ Andrews Experimental Forest? Insert a snapshot with the streams and roads that are inside HJA Andrews.***

Question#4 [10%]: Measurements.

In many applications, you are interested in finding the size of the various features, such as length or area. Using the stream and road layers **inside** of the HJ Andrews Experimental Forest, you are tasked to find the total length of the stream network. You will accomplish this task in tow steps: first, you will create a field in the table called Length_m (meaning length in meters), and second, by using the Calculate Geometry option available when the database of the shapefile is open. ***What is the total length of the streams and roads network in kilometers, not in meters?***

Question#5 [10%]: Buffers.

In many instances, you would like to create a buffer around linear features, such as the streams from HJ Andrews. In ArcGIS, there are two options on how to create the buffers: as one unit or as individual entities, as defined by each feature from the input layer. This option is under the Dissolve Type window.

Create two buffers for the streams inside HJ Andrews with a size of 50 m on both sides; one with individual features and one with only one feature (all features merged).

What is the total area of the two buffers? Why do you think they are different?

Question#6 [20%]. Intersection.

In many applications, you are interested in an area that is common to different processes or properties. In this exercise, you are interested in finding the chance of road-related erosion into streams based on their relative position. You could assume that a road located further than 50 m from a stream would have a minim amount of sediments reaching the stream; therefore, you will intersect the streams buffer layer created in the previous exercise with the roads layer. The resulted file should be a line feature.

Which buffer filed should you use: the one with individual features or the one with all features combined (merged)?

What is the length of the roads that poses a sedimentary risk?

Besides the length that possesses the risk to water quality, you could be interested in the area that is more susceptible to receive the road sediments. To identify the respective area, you could create a buffer around the roads layer (also 50 m wide), then intersect it with the buffer of the streams layer. The resulted file would be a polygon feature.
What is the area likely to receive the sediments from the roads? Include a snapshot with the common area. Is this approach accurate?
Mention at least one issue posed by the proposed method of identifying the areas that could receive sediemnst from the roads.

1.6 Operations with raster data

Question#7 [5%]: Information within the raster
In most instances, when working with rater data, it is important to know the spatial resolution of the image, namely the size of the pixel.
What is the spatial resolution of the HJA DTM and the HJA DSM rasters?

Question#8 [10%]: Crop an image
In many situations, you are interested in cropping an image according to a polygon, such as a DTM to a watershed boundary. You are tasked to crop the HJA Andrews DTM, which is for a larger area than the Experimental forest, to the boundary of the HJ Andrews Experimental Forest. You can execute this task by using the Clip function from the Data Management→ Raster → Raster Processing folder.
How many pixels are located in the clipped raster?
What are the minimum and maximum values present in the clipped raster?

Question#9 [10%]: Coloring a raster
Many images, such as panchromatic images, are not very appealing visually. Therefore, you want to create a nicer image. For rasters, this process is very simple if just limited displays are required. If you double click the name of the raster file in the Table of Contents, the Layer Properties will open, from which you should choose the Symbology Tab. From the available color ramps, choose one that you think would produce nice images (like in Figure 4).

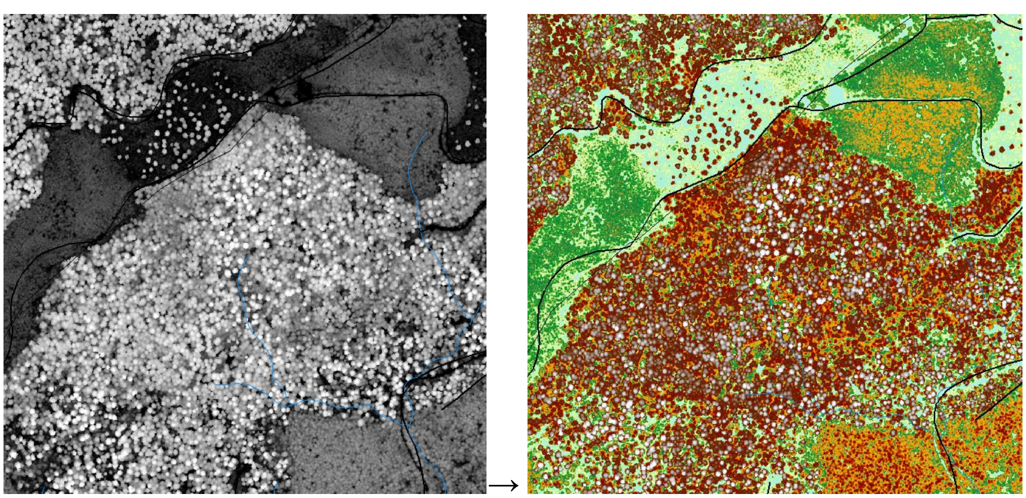


Figure 4. Change in the display of information for raster data.

Produce an image that uses a color ramp that is visually pleasing but does NOT contain the white color.

Question#10 [10%]: Identify cells above a particular value.
Rasters are famous for being able to be analyzed with ease. Therefore, you would like to perform some statistical analyses on the rasters. One of such simple statistics could be areas above a particular value. This question is valid in levies and dikes assessment or tree identification.In ArcGIS the answer to this question can be obtained using Map Algebra.

In this question, we are interested in finding tall trees, whose crown can be found using the DSM layer. Among a large number of options available in Map Algebra that can answer this question, we will be using the Conditional function accessed thru the Raster Calculator. The function **Con**, assign the value 1 when the pixel value is larger than 200 and 0 otherwise. The syntax of the function is ***Con(condition, value when the condition is true, value when the condition is false***). In our particular case, the function looks like this:

Con("HJA_DSM.tif">200,1,0)

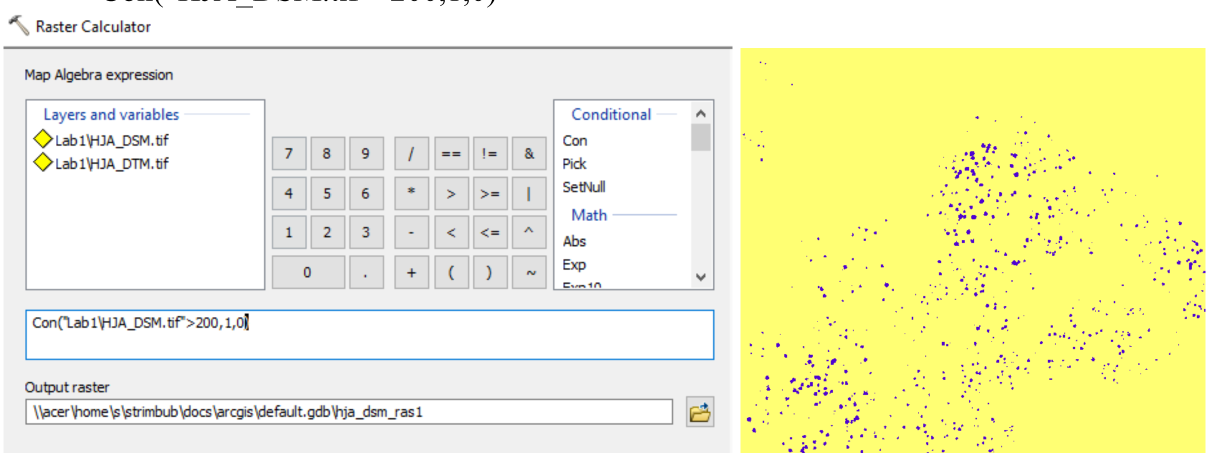


Figure 5. The Raster Calculator interface (left) and the results of the Conditional function (right)

How large is the area covered by large trees? You should answer this question by multiplying the size of a pixel with the number of the pixels that have the value 1.

1.7 Advanced topics: Introduction to ERDAS IMAGINE

1.7.1 Objective

The objective of this lab is to familiarize you with ERDAS IMAGINR interface, also referred to as the workspace, and execute some basic image manipulation and information extraction with ERDAS IMAGINE.

1.7.2 Files Used in This Lab

The files used in Lab#1 are located on Canvas under the Week#1 module. They are associated with the Landsat 5 image of the Gulf Islands from the Strait of Georgia acquired in 2006, and are presented as ENVI file:

- LandsatTM.hdr is header file contains metadata for the .dat file.

- LandsatTM.dat is the file that contains the actual image, but it cannot be display without the hdr file.

1.7.3 ERDAS

1.7.3.1 Overview

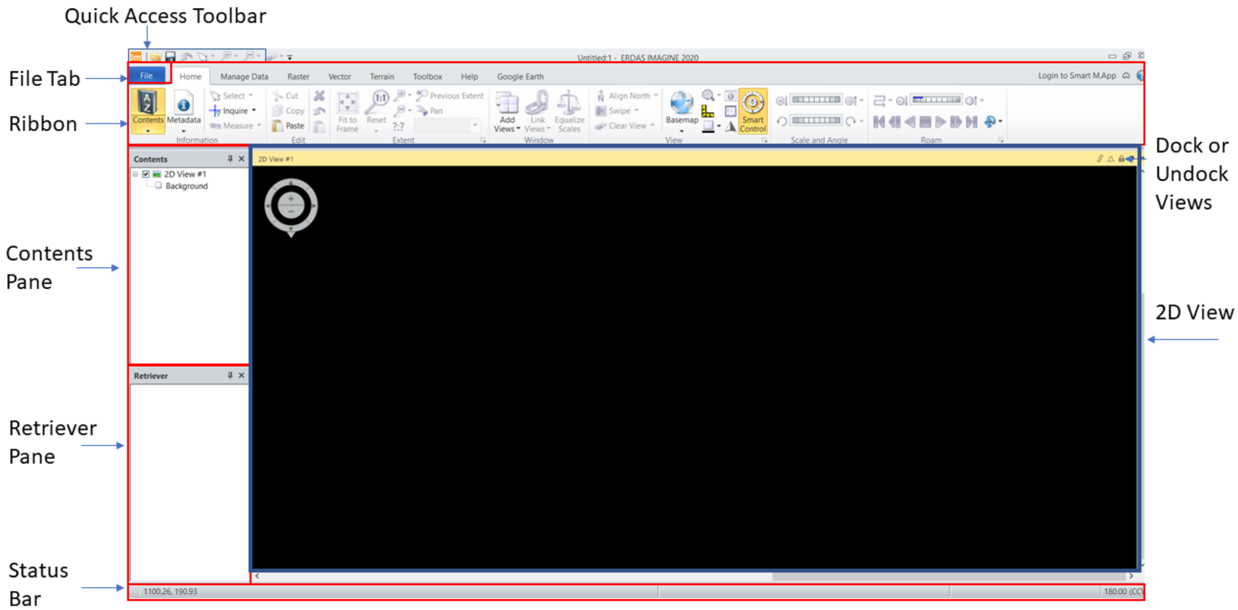
Hexagon Geospatial, or Hexagon GSP, claims that ERDAS Imagine is the “world’s most widely-used remote sensing software package.” ERDAS IMAGINE is a suite of software that bundles remote sensing, Photogrammetry, LiDAR analysis, basic vector analysis, and radar processing under one umbrella. ERDAS IMAGINE is a raster-based software designed to extract information from images. Unlike ENVI, another popular ERDAS for education is equipped with a series of additional packages, which are very important in forestry applications, particularly:

- ERDAS IMAGINE Professional, which is a graphical Spatial Model Editor for building and executing re-usable spatial recipes, multispectral image classification, hyperspectral image processing, and point cloud tools;
- IMAGINE Expansion Pack, which includes automated image-to-image co-registration (AutoSync), 3D visualization and analysis (VirtualGIS), wizard-based change detection (DeltaCue), orthorectifying Radar data (OrthoRadar), NITF support, extracting terrain from SAR images, and stereo feature collection;
- ERDAS ER Mapper, which has advanced image processing and compression capabilities geared toward oil, gas, and mineral industries;
- IMAGINE Photogrammetry, which includes Stereo and multi-image eATE; and
- IMAGINE Terrain Editor, which includes TE and Stereo.

In this class, we will be using mainly Imagine and Photogrammetry programs from the available suite of programs.

1.7.3.2 Imagine interface

To start ERDAS application simply press the Windows icon on the screen, then type ERDAS. From the available options that will appear choose ERDAS IMAGINE, which will open the user interface. The interface contains various panels that allow you to visualize and organize your datasets. The basic layout of the ERDAS Workspace contains several major components, such as **Quick Access Toolbar, Ribbon** (with various tabs, such as File Tab)**, Contents pane, Retriever pane, or 2D Viewer,** some of which are briefly explained below and presented in the next figure.

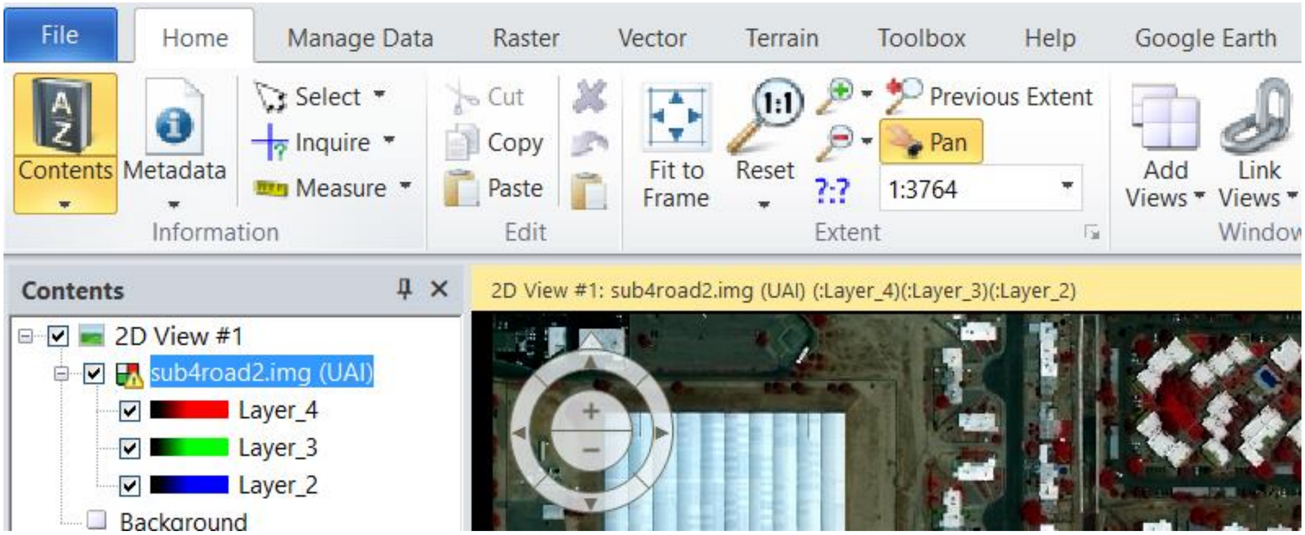


Quick Access Toolbar includes tools commonly used in ERDAS IMAGINE, such as save, undo, zoom in, zoom out, etc. It is usually at the top of the screen, right beside the Im icon:

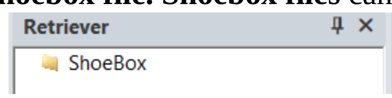
The **Ribbon** contains all the major functions of ERDAS IMAGINE, which can be accessed thru the Manu Bar and the Tool Bar. Inside the **Ribbon**, you can see several tabs (e.g., **Home, Manage Data, Raster**, etc.), and each tab contains groups (i.e., groups such as **Information, Edit, and Extent** located inside the Home tab). Furthermore, you can find a set of tools inside each group. Depending on the active **Layer** in the Contents pane, you may see changes of tabs.

The Menu bar provide access to all the functions available in ERDAS Imagine. There are many options form the Menu bar, such as the **File tab, Home, Raster, Terrain or Toolbox.** The File tab provides basic functions that allow the user to open layers, save layers, set user preferences, configuration, print, etc. The Home tab is the main interface of ERDAS Imagine, as it provides the viewer as well as the basic functionality of data inquiry. The Raster, Terrain or Toolbox tabs are similar with the Home tab, in the sense that the viewer is still maintained, but the Ribbon changes to provide access to the main functions related to the respective tab. One detail that should be noticed, is that the lidar options, such as point classification, are found under the Terrain tab.

The **Contents pane** hosts the data you have opened in the workspace and Viewers (i.e., 2D View#1).

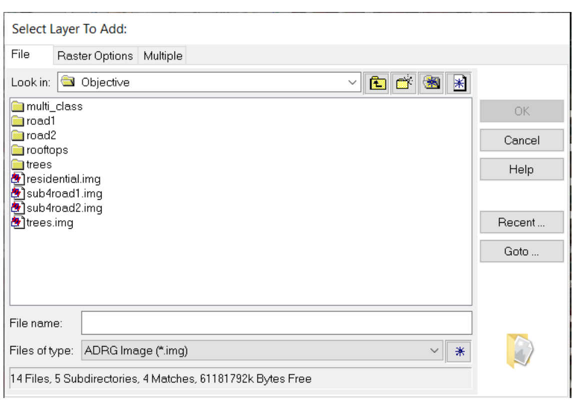


The **Retriever pane** gives quick access to the data by creating a **Shoebox file. Shoebox files** can be made by right-clicking on the Retriever pane. Once you added the data into Shoebox files, it allows you to drag-and-drop data from the Retriever to the Viewer.



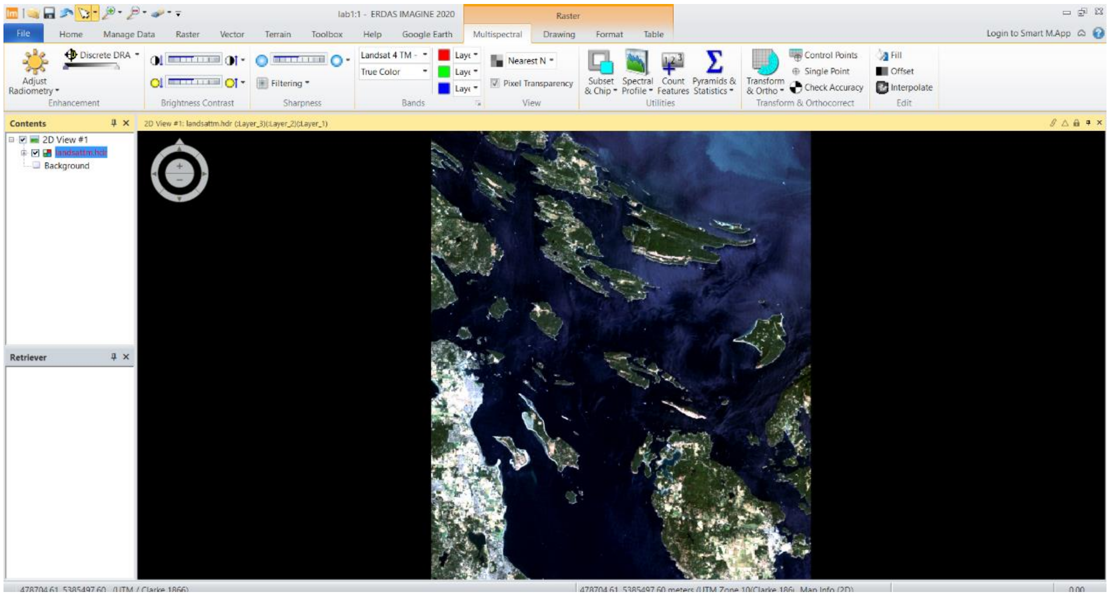
1.7.3.3 Display Images

1. From the File tab, select **File > Open> Raster Layer**. A file selection dialog appears. Navigate to the data folder by using the **Look in** drop-down menu. Also, check and change the file type as you need. Usually, it is better to select "**All File-based raster Formats**" option in **Files of types** drop-down menu.

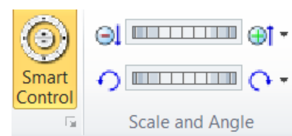


2. After the navigation to the data folder hosting the Landsat image, select the file `LandsatTM.hdr`. Then click **OK.** A simpler option of loading the data is by dragging and dropping the file in the Viewer.

Note: If you want to make the current directory as the default directory, you can click  icon (third icon from left). Also, you can select a default output folder by clicking  icon (first icon from right).

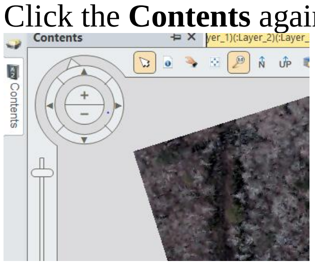


3. Once the image is loaded, go to the **Home tab**, click **Zoom In** icon  to zoom in, and **Zoom Out icon**  to zoom out. Alternatively, you can use the **Smart Control** option to do the same tasks (i.e., Zoom In, Zoom Out, Pan, etc.).



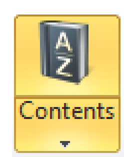
4. **Scale and Angle** tool, which works like a dial rotating perpendicular to the screen, provides an easy option to change the size of the image and angle of the image.
5. Hide the **Contents pan** by clicking AutoHide button .

6. Click the **Contents** again to expand it.



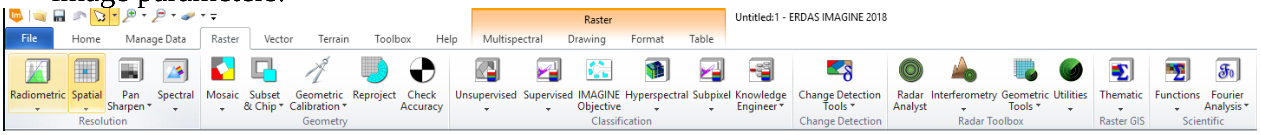
7. Also, you can drag the two pans if you need to detach them from the user interface. They become floating dialogs that can be moved to a second monitor, for example.

8. Clicking the X button in the upper-right corner of the **Contents pan or the Retriever pan** will remove those pans from the main interface. To turn them back on go to Home tab, then under the Contents icon select the pan that you just closed.

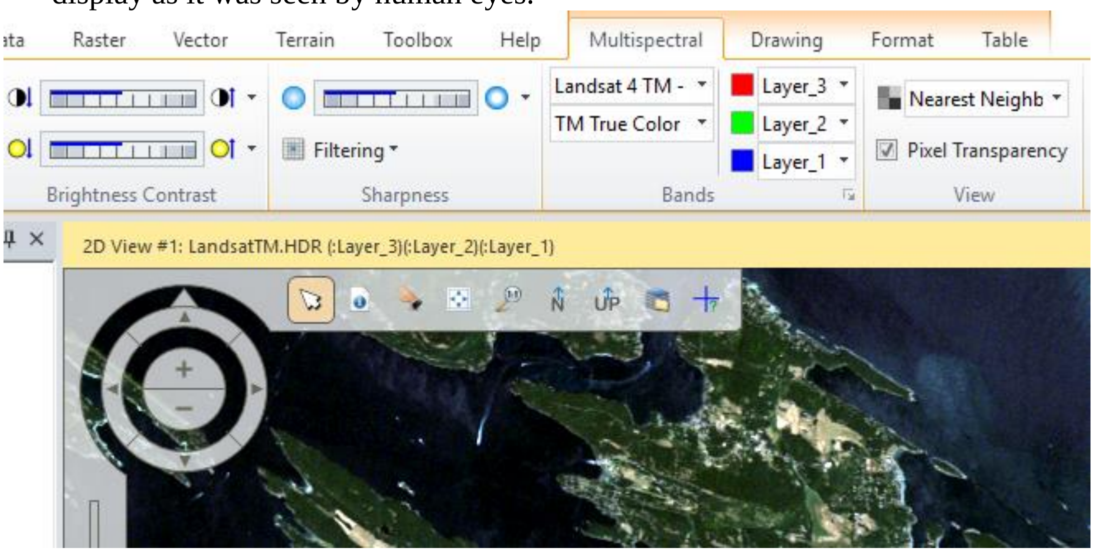


9. To navigate inside the image, click the **Pan** button in the toolbar, then click and drag on the image to move around. You can also use the middle mouse wheel to zoom in and out of the image. If you need to fit the image into the window, you can right-click in the image in the **Contents pan** and select **Fit Layer to Window** option.

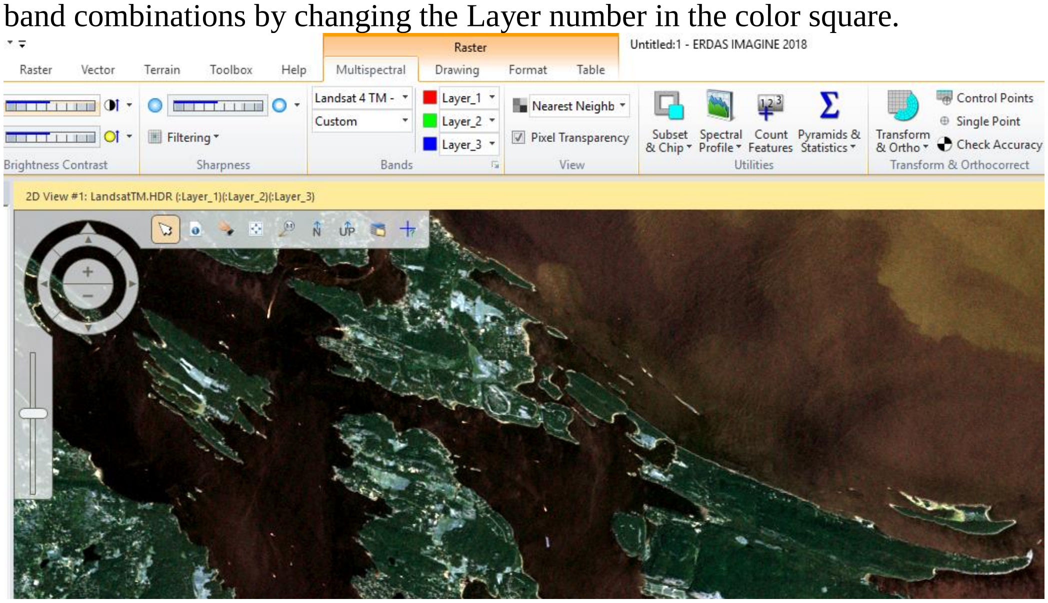
10. From the Ribbon, select the **Raster tab** and then the **Multispectral tab** to adjust the image parameters.



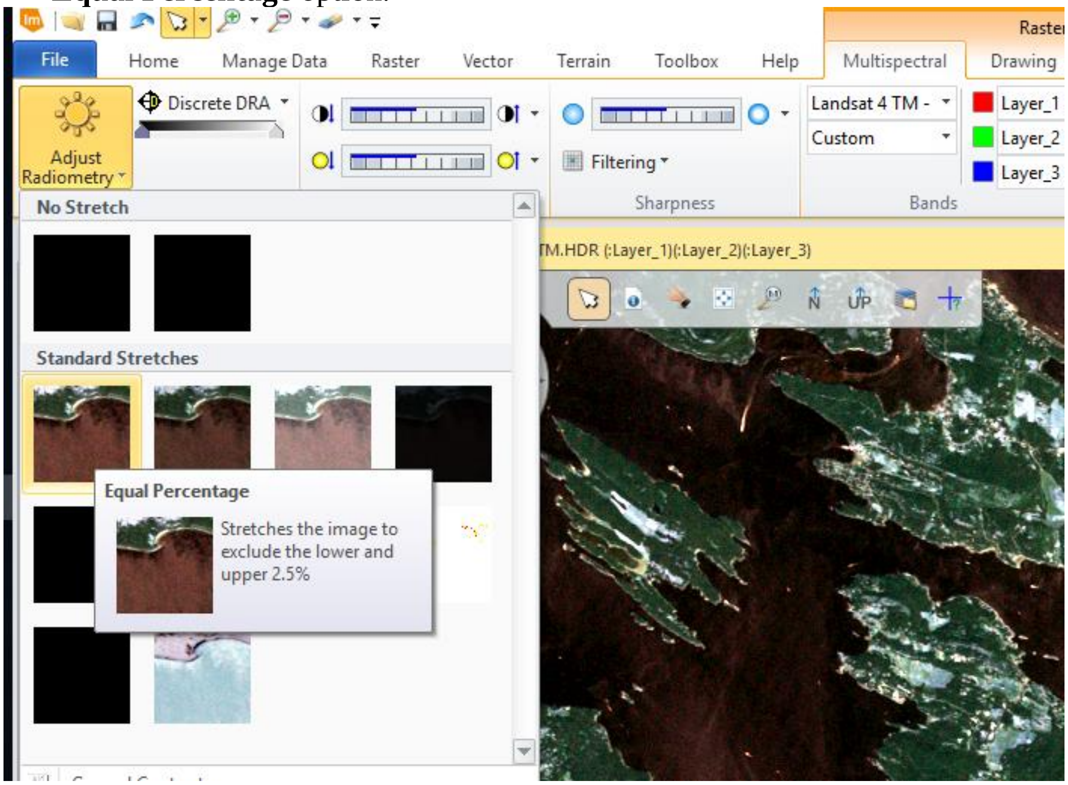
11. The **Bands group** allows you to use the spectral reflectance of the selected image. If the sensor is known it will be identified in the **Sensor Type.** If the sensor is identified, you can change the band combination by using the drop-down menu from the **Band combination**. Select TM True Color, and the image would be display as it was seen by human eyes.



12. Select Custom from the drop-down menu and select in order **Layer 1**, **Layer 2**, and **Layer 3,** for **red, green, and blue color square**, respectively. Notice that each selected Layer has a color square to its left in the Bands tool. You can create different band combinations by changing the Layer number in the color square.



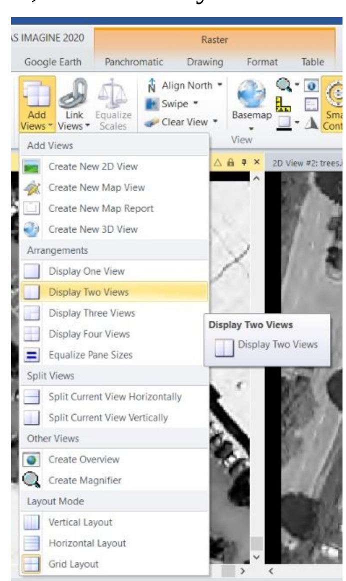
13. **Adjust Radiometry** potion provides different stretches to the original image. Select the **Equal Percentage** option.



14. Change the brightness, contrast, and sharpness appropriately using the **Brightness Contrast and Sharpness** tools. Click in the **Filtering** option and select the **Sharpen** Filter option.

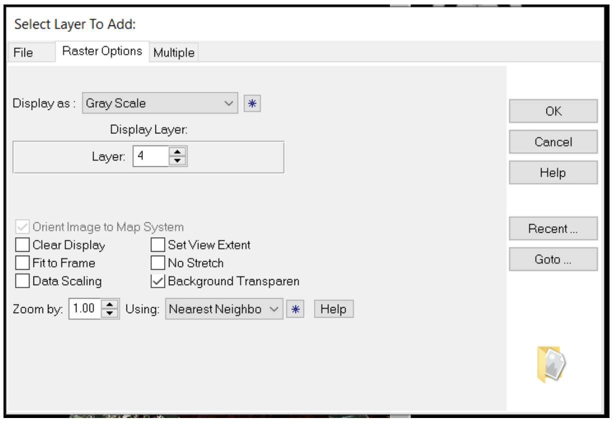
1.7.3.4 Multiple Views

At this point, the Contents pane has a single image. If you want to work with multiple images, which most likely will be the case, in ERADS you have two options. One, is like in any GIS software: have all the images in one place, which in ERDAS will be the Contents pane. However, in many remote sensing specialized software, like ERDAS, you have another option, namely images displayed in different viewers. Instead of overlaying layers and toggling them on and off in the Contents pane, you can create multiple views to compare images side-by-side using the **Add Views** option.

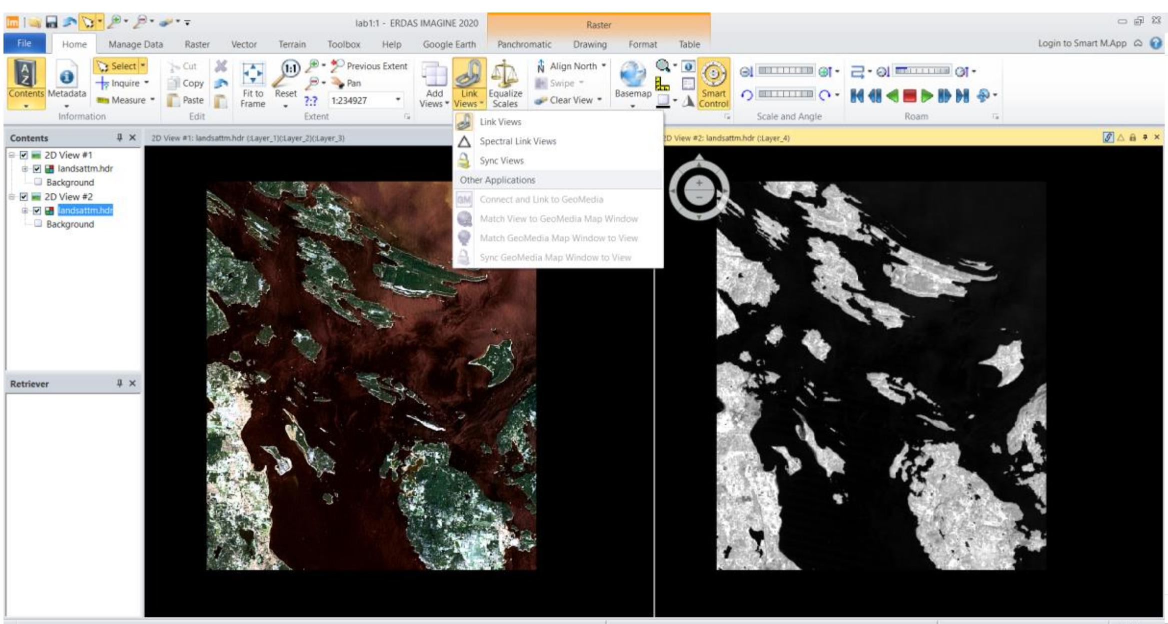


1. From the menu bar, select **Home** tab and click **Add Views > Display Two Views**. A new view appears to the right of **2D View#1** window. You can see the newly added view in the Contents pan as **2D View#2**. Also, **2D View#2** is the *active view*, highlighted with a pale-yellow colored border in the Image window. All subsequent navigation and image enhancement operations will be in the active view.

2. In the Contents pane, right-click on **2D View#2** and open the **LandsatTM.hdr** file in the viewer. Ten select **Raster Options** tab. Click on **Display as** drop-down menu select **Grayscale option**. This will display only one band out of the six available. If you want to display only the band 4, then select 4 from "Display Layer" option.

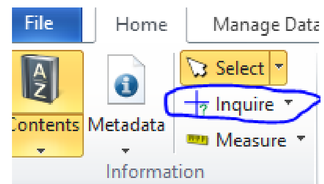


3. A Gray Scale layer is added to the second view. Right-click on the image under 2D View#2 and select **Fit Layer to Window.**
4. Because **LandsatTM** is georeferenced to a standard map projection, you can link the views by geographic location, meaning the images in the two Viewers would be synchronized. From the Home Tab, select the **Link Views option (next to Add Views option)**.
5. To keep both Views on the same band scale, click the **Equalize Scales** option. Also, you can try other Link Views option as well to get familiar with different link views. Especially, **Sync Views** option is useful to move both the images together.

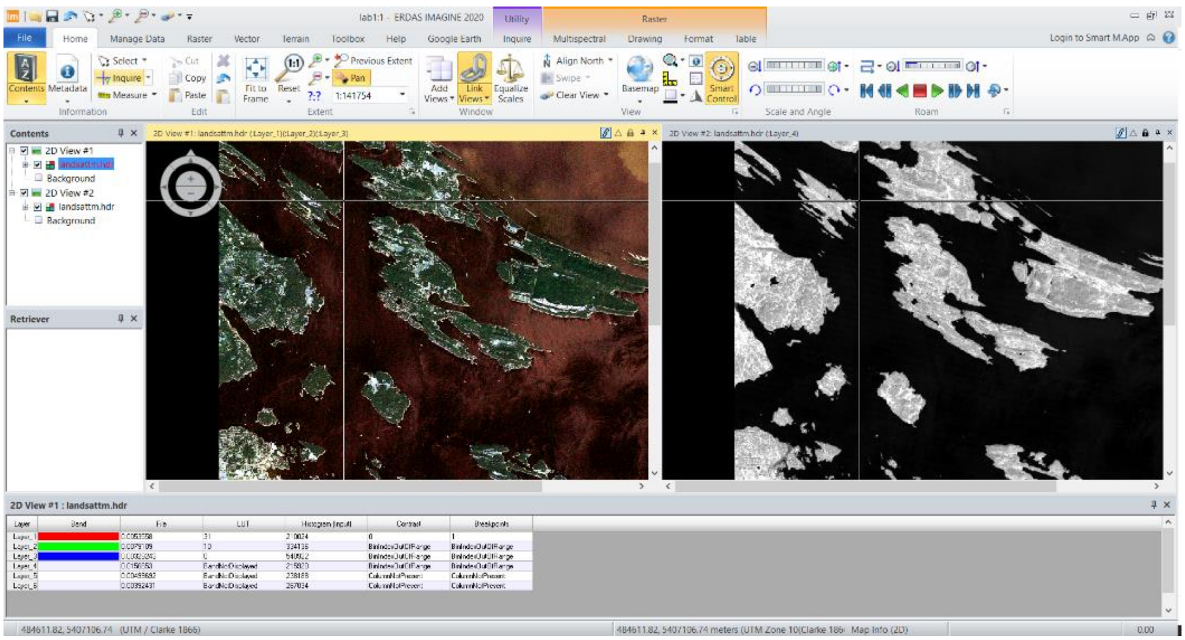


6. Click the **Pan icon** in the toolbar and pan around the grayscale image. The color image moves as well, and the same area is shown in both views.

7. Go to the **Home tab, Information group**, then select the **Inquire option**, to open **Inquire Cursor utility**. Now you can see a cross-hair in both the Views. Also, note in the embedded panel (at the bottom of the window) the pixel values for the currently active image in the **Contents pane** (at the location of the Inquire Cursor cross-hair displayed).



This Data table shows the red, green, and blue digital number (DN) values for the active view. If you want to see the data for both Views, you can select Show All Layers option as well.



8. Once you select the **Inquire Cursor utility, Utility Inquire Tab** will open. In the coordinate group, you can see the coordinates for the selected pixel.

9. Alternatively, in the **Inquire tab**, you can select the coordinate option to enter the latitude and longitude values of the pixel that you are interested in. Also, you can use **Move group** and **Move Cursor Up/Down/Left/Right** options to move the cross-hair.

10. Select the **Close Inquire** option to exit.

1.7.3.5 Image Meta-data and Histograms

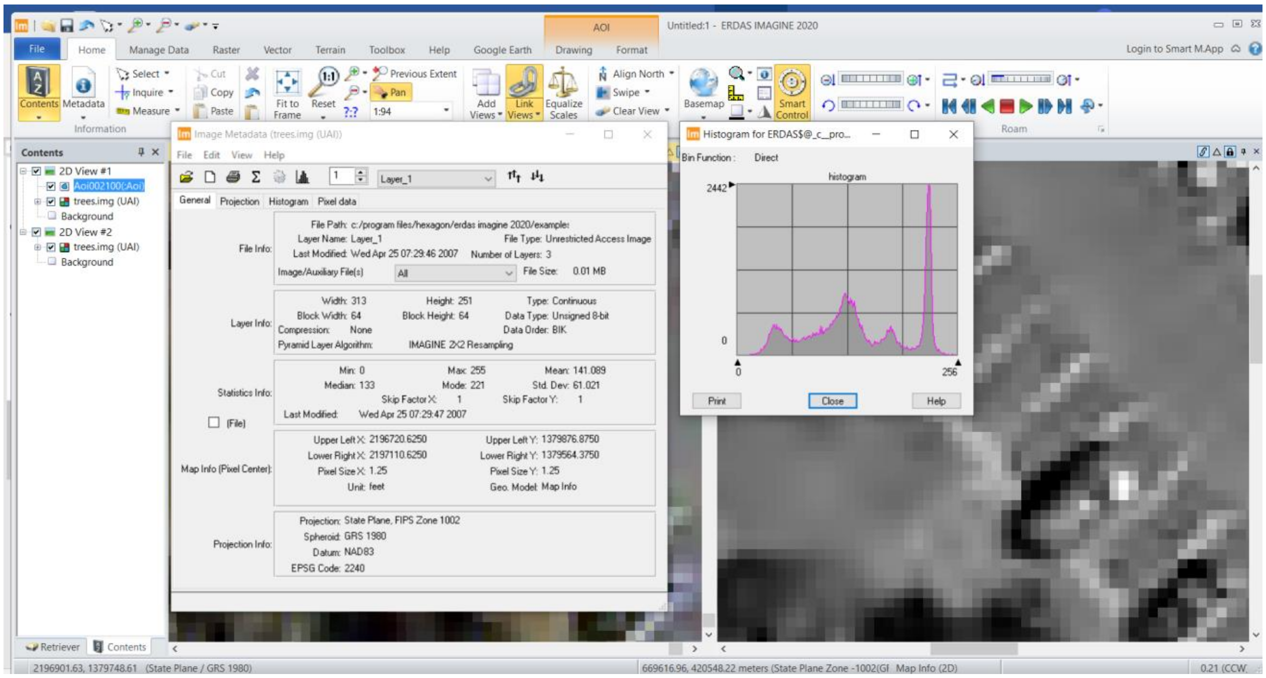
These options provide details of Image Metadata, such as image projection and spatial resolution.

1. Select **View#1** to activate the **color image**. Go to **Home Tab** and click the Metadata option. This will open an **Image Metadata dialog box** that has **General information, Projection information, Histogram, and Pixel Data.**

2. Click on **Histogram Icon or Histogram tab**. This will open a histogram where the X-axis displays the range of possible brightness values, and Y-axis displays the number of

pixels for any brightness values. By changing the Layers in the **Image Metadata dialog box**, you can observe respective histograms for each Layer.

3. Bring the cursor to the top of the histogram; now, you can see the X and Y values respective to the courser point.
4. Select the **Pixel data tab** on the **Image Metadata dialog box.** This option will allow you to see the pixel value for the selected Layer.



1.7.3.6 Saving the Workspace

1. To save the session, select **File** > **Save As** > **Session**.
2. This allows you to save the entire workspace, including open Viewers, data, and view extents.

2 The Electromagnetic Spectrum, ArcGIS and ERDAS

2.1 *Objectives*

- Usage of reflectance in identification of land features
- Usage of Landsat images
- Labels in ArcGIS
- Annotations in ERDAS

2.2 *Description of the files*

- Quickbird from Vancouver BC: Quickbird.tif is a QuickBird multispectral image of Vancouver BC
- QuickBird from Colorado image: qb_colorado.dat is a QuickBird multispectral image of Boulder, CO
- QuickBird from Colorado header: qb_colorado.hdr is the header file of the above QuickBird image

2.3 *Overview*

The electromagnetic spectrum is the distribution of electromagnetic radiation according to wavelength/frequency, and includes radio waves, visible and infrared light, x-rays, gamma rays, and more. In remote sensing, we use the reflective, absorptive, and emissive properties of terrestrial features to identify and measure them. Below is a series of questions. When answering these questions, you do not need to cite your references, but *do not copy and paste*! *Put things into your own words*. Copying and pasting/not putting things into your own words will result in point deductions.

Q1) *Define "atmospheric window."*

Q2) *Define the three types of atmospheric scattering. Compare and contrast them.*

2.4 *Spectral Reflectance*

Figure 1 shows the reflective characteristics of various materials. Answer the following questions.

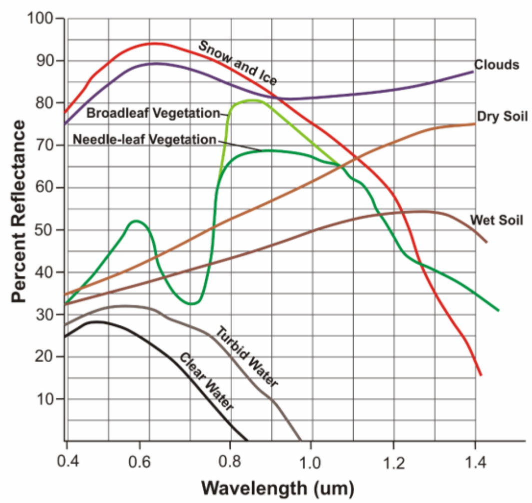


Figure 1: Reflectance characteristics of various features at different wavelengths.

***Q 3)** What wavelength range has the highest reflectance for snow and ice?*
***Q4)** In which regions of the spectrum can dry soil be best distinguished from vegetation?*
***Q5)** What is the reflectance of water at 1.2 µm? What would it look like if we could see at this wavelength?*
***Q6)** From this graph, what color do we think clouds are? This is not a trick question…*
***Q7)** Where is the greatest differentiation in reflectance for the two vegetation types?*
***Q8)** Why is there no separation between vegetation types at 0.6 µm?*
***Q9)** What is water turbidity? Why are there differences between the clear water vs. turbid water spectra?*
***Q10)** What wavelength region has the largest change in reflectance for vegetation (in other words, where is the curve steepest)?*
***Q11)** List the similarities between the dry and wet soil spectra compared with the clear and turbid water spectra.*
***Q12)** Which cover types look almost identical at 0.85 µm (+/- 10%)? At 0.4 µm (+/- 10%)?*
***Q13)** Use the wavelength and percent reflectance data in Table 1 to plot reflectance curves onto Figure 1. These curves can be plotted using MS Excel. However you want to do this is fine.*

Table 1: Spectral characteristics of five unknown features.

Wavelength (µm)	Spectra 1 (%)	Spectra 2 (%)	Spectra 3 (%)	Spectra 4 (%)	Spectra 5 (%)
0.40	17	30	34	32	80
0.45	18	31.5	37.5	34	80.5
0.50	15	33	42	36	81
0.55	12	34.5	44	38	81.5
0.60	5	35	43	37	82
0.65		35.5	40	35	82
0.70		35	37	36	82
0.75		34.5	38	43	82
0.80		34	46	49	82
0.85		33	59	53	82.5
0.90		32	67	55	83
0.95		31	70.5	53.5	82.5
1.00		29	71	51	82
1.05		26	70.5	46	81.5
1.10		22	70	41	81
1.15		17	69.5	36.5	81
1.20		14	69	33	81
1.25		10	70	31	81
1.30		6	71	29	80
1.35		0	72	27.5	80
1.40			72	25	80

Q14) *Identify the five features that are represented by the spectral data in Table 1. Make an educated guess based on the known reflective properties of various earth surface materials and what you have learned from class/your readings, and explain your reasoning. Take a look and compare to the spectra seen in figure 1. This is a difficult task, and educated guesses are all that is asked for. You are not expected to get all of these right. However, do a bit of research and put some thought into it. Use the curves already identified as your starting point.*

2.4.1 Landsat 7 Bands and the Electromagnetic Spectrum

Table 2 Parameters of Landsat 7's enhanced thematic mapper (ETM+) sensor.

Band	Spectral Range (microns)	Spectral Region	Spatial Resolution (meters)	Applications
1	0.45-0.52	Blue	30	Coastal water mapping, differentiation of vegetation and soils.
2	0.52-0.60	Green	30	Assessment of vegetation vigor.
3	0.63-0.69	Red	30	Chlorophyll absorption for vegetation differentiation.
4	0.76-0.90	Near Infrared	30	Biomass surveys and delineation of water bodies
5	1.55-1.75	Middle Infrared	30	Vegetation and soil moisture measurements. Differentiation of ice and clouds.
6	10.40-12.50	Thermal Infrared	60	Thermal mapping, soil moisture studies, plant heat stress measurement.
7	2.08-2.35	Middle Infrared	30	Hydrothermal mapping.
8	0.52-0.90	Green, Red, Near Infrared	15	Panchromatic band. Large area mapping, urban change studies.

Figure 2 shows individual Landsat 7 ETM+ channels in grayscale. Notice how the reflectance characteristics of various features (e.g. urban areas, water and ocean sediment, forests) changes with different wavelengths.

In a true color image, the computer display represents each band using the same color that the sensor measured when the data was acquired. In other words, in a true color image, Landsat band 1 (blue) is displayed through the computer monitor's blue gun, band 2 (green) is displayed through the computer monitor's green gun, and band 3 (red) is displayed through the computer monitor's red gun. Any combination where this is not the case (which includes any other combination), is a false color composite.

In false color composites, image data are displayed using wavelengths other than the wavelengths that were measured. False color composites are necessary because human sight can usually only perceive the visible portion of the spectrum (of course!), while sensors on remote sensing platforms can measure a much broader range of wavelengths than humans can see. As a result, in order to present these data visually for humans, they must be assigned to the part of the spectrum that humans can see.

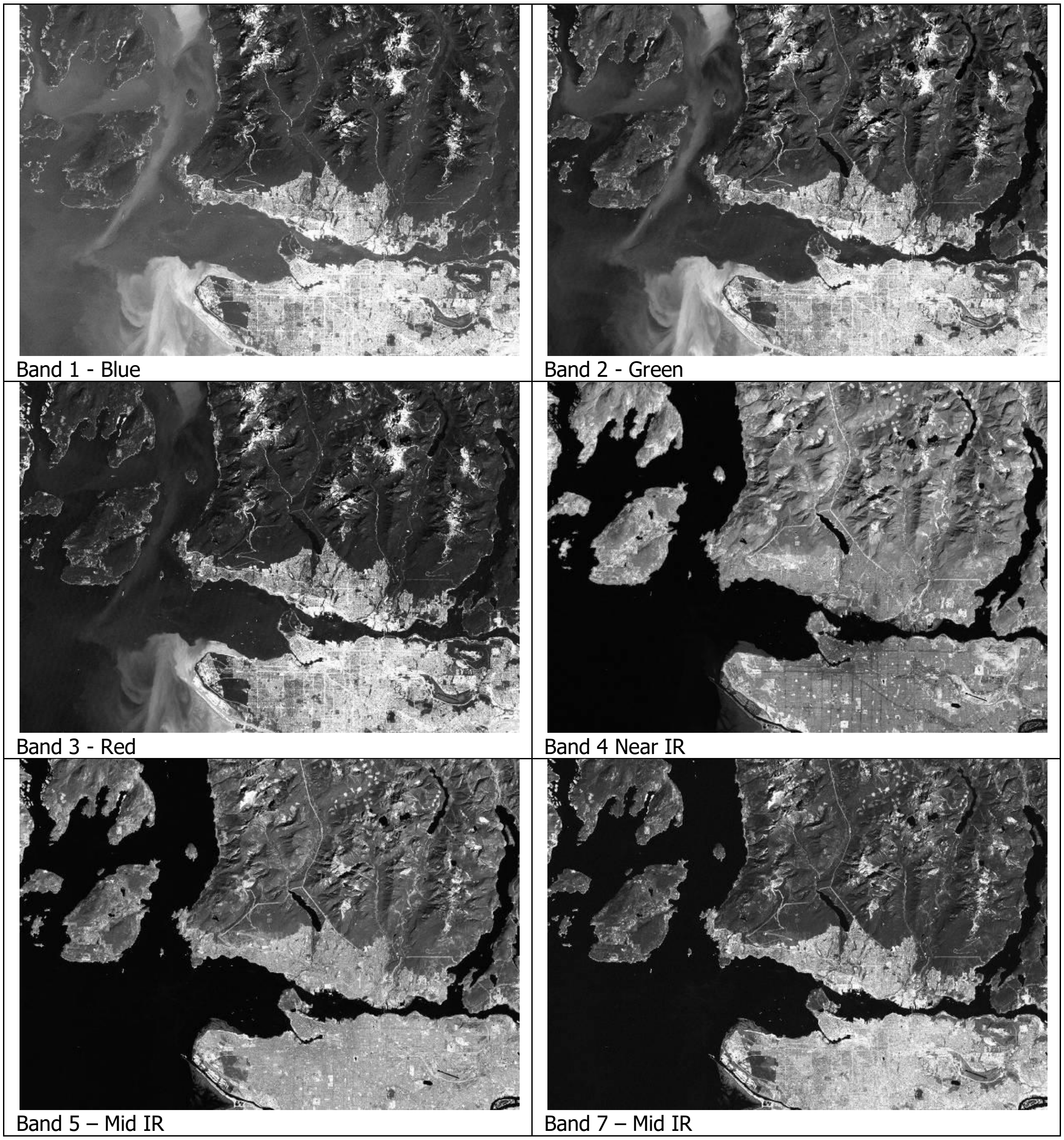


Figure 2: Individual Landsat 7 bands shown in gray-scale.

Figures 3a to 3g display color composites of Vancouver created from Landsat 7 ETM+ sensor data. For each composite, the order of the Landsat band number indicates which color gun it is being displayed through (red, green, and blue). All composites follow the same color order, be it RGB = 321, RGB = 432, RGB = 543, etc., this just means that this first number (which represents the corresponding Landsat band (i.e., 1-7) is being represented by the color red in the image, the second number (Landsat band) is being represented by the color green, and the third number (Landsat band) is being represented by the color blue.

Examine Figures 3a to 3g and answer the following questions, keeping in mind the relationship between the electromagnetic reflectance and absorbance of terrestrial features and the color gun representations.

Q15) Why does vegetation appear red in Figure 3b, but blue in Figure 3g.

Q16) Which composite is best for mapping ocean sediments?

Figure 3a: RGB = 321 true color composite.

Figure 3b: RGB = 432 standard false color composite.

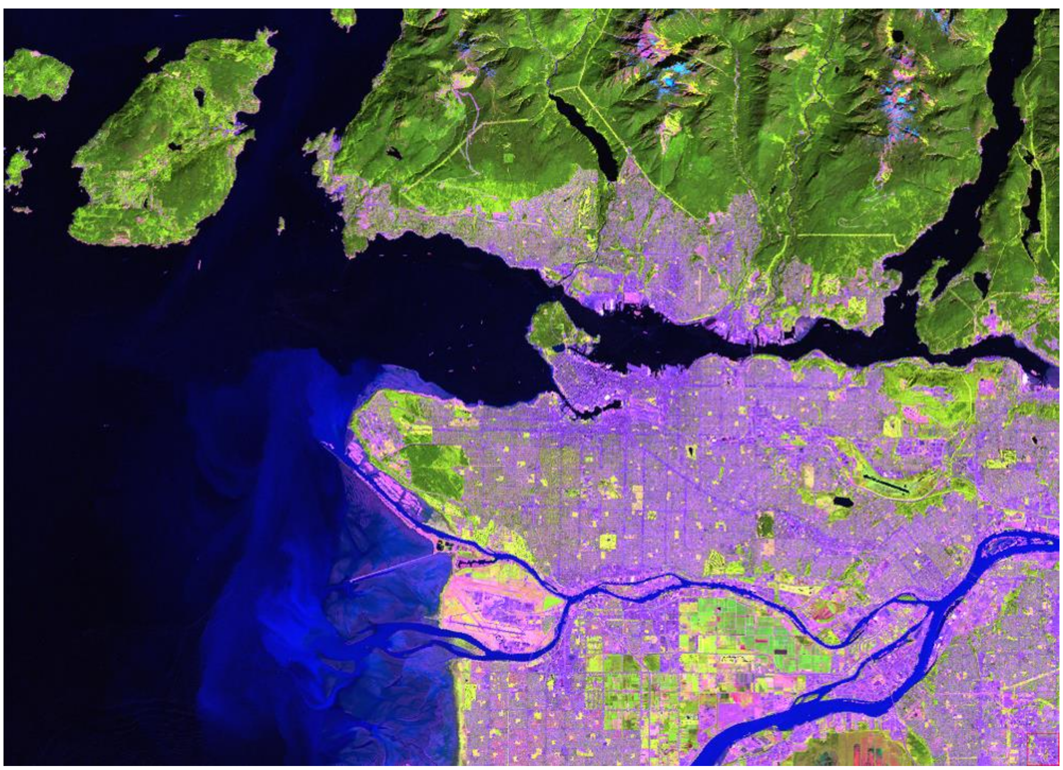


Figure 3c: RGB = 543 false color composite.

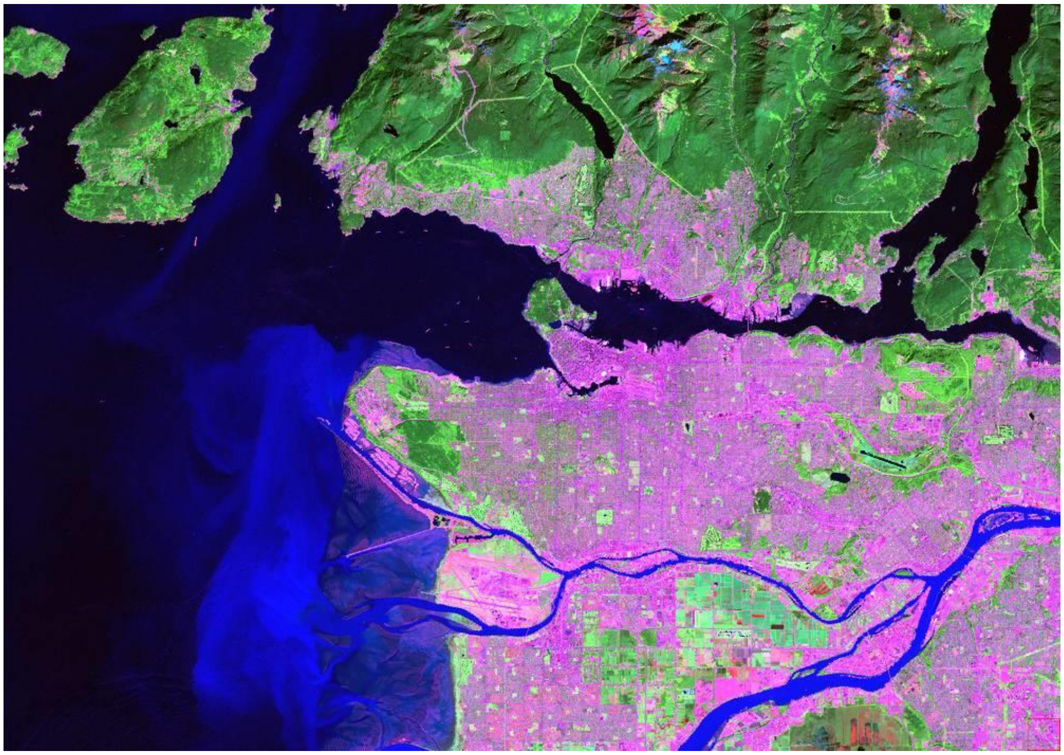


Figure 3d: RGB = 742 false color composite.

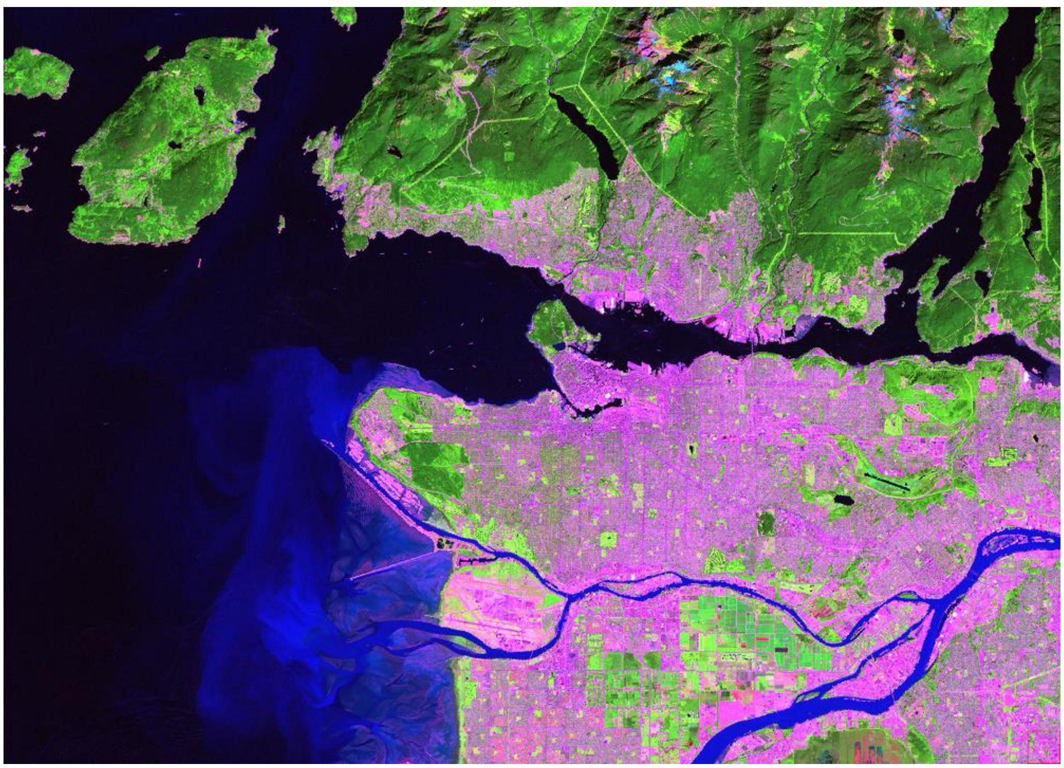


Figure 3e: RGB = 743 false color composite.

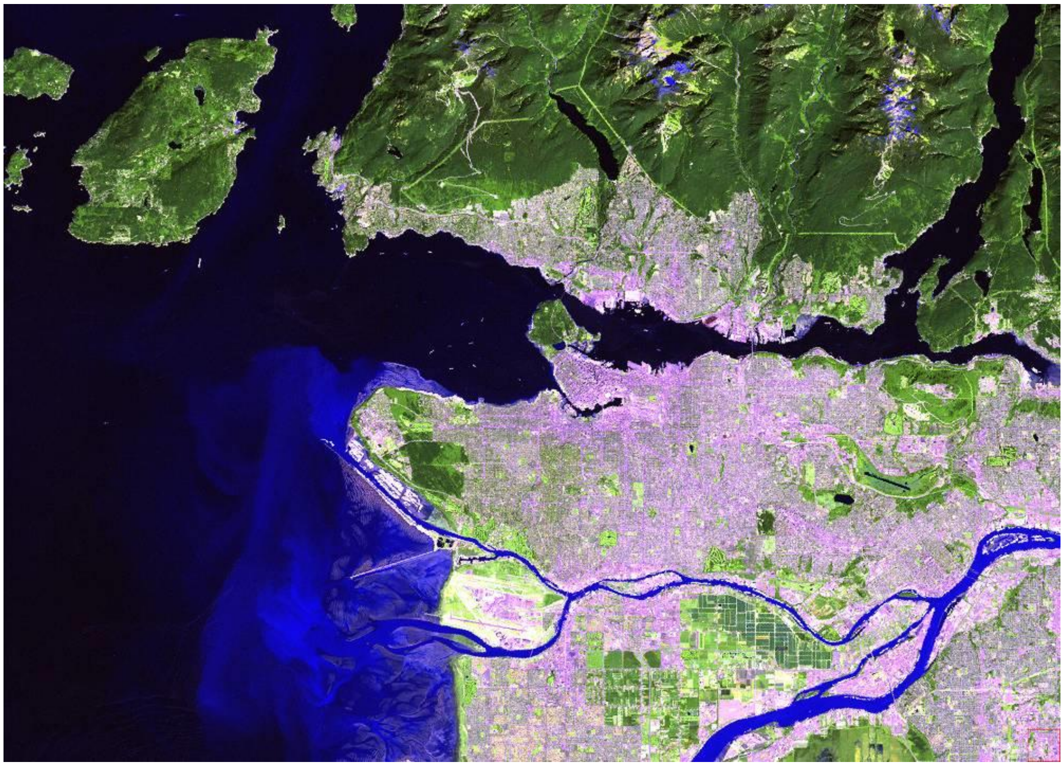


Figure 3f: RGB = 753 false color composite.

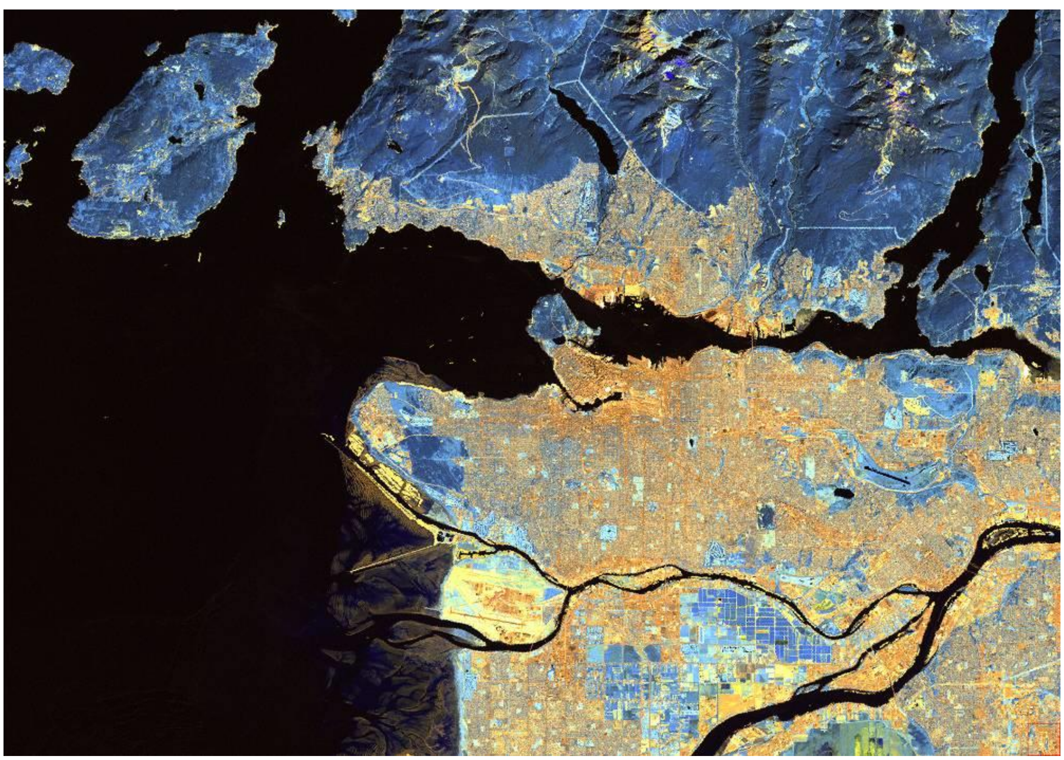
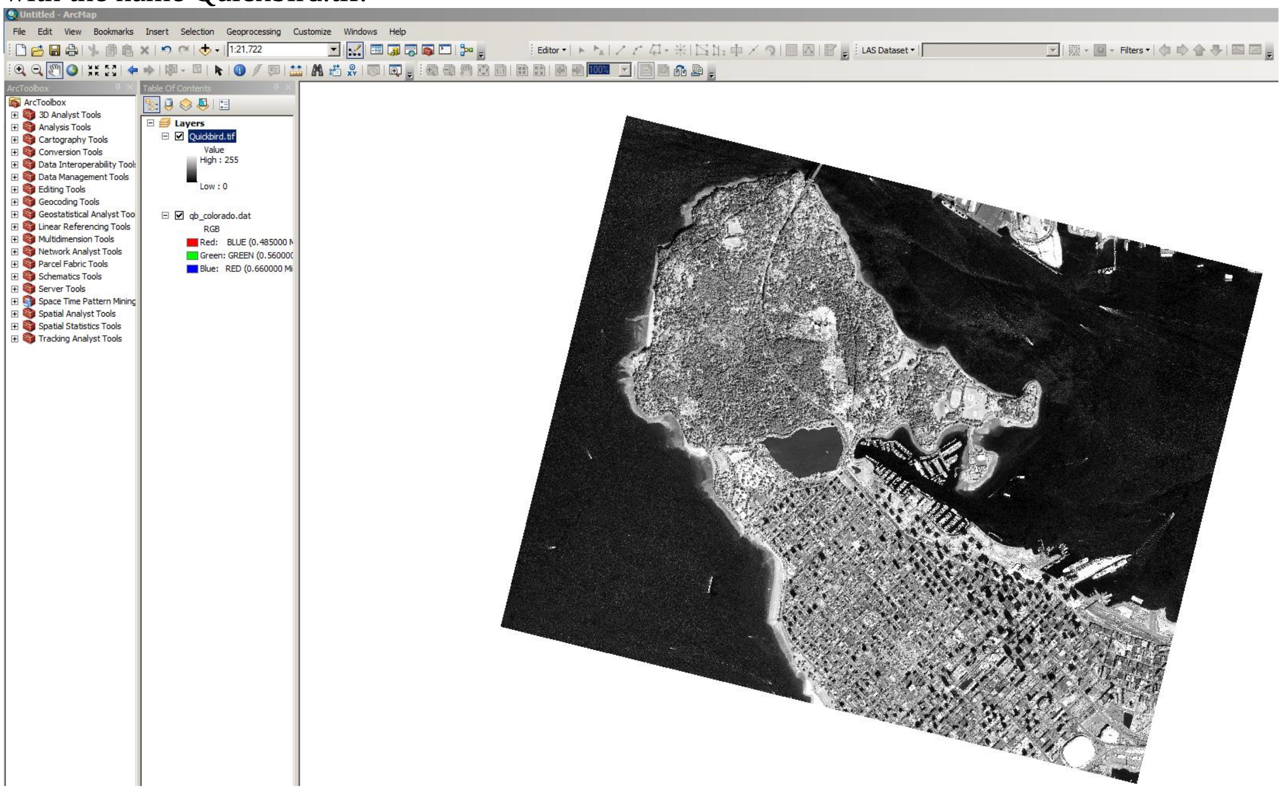


Figure 3g: RGB = 754 false color composite.

2.5 *ArcGIS*

2.5.1 Basic image analysis

ArcGIS, developed by ESRI, is the de-facto software used in geospatial manipulations. ArcGIS works with both vector and raster data, and lately with point clouds. Therefore, it is very important for you to be familiar with the features that ArcGIS has, particularly in the Remote Sensing area. We will start by uploading a Quickbird image of the city of Vancouver BC, which has all three geomorphologic features (planes, hills and mountains), as well as three types of water bodies: ocean (salty), river and lakes (fresh water). The image is a panchromatic tiff file with the name Quickbird.tif.



Look up the Quickbird sensor using Google and briefly read about its characteristics. Answer the following questions:

Q17- This image is panchromatic, what does that mean?
Q18- What is the spectral range of the QUICKBIRD panchromatic band?

Q19 – What is the spatial resolution of the image? You could find this information in the Source tab of layer's properties.

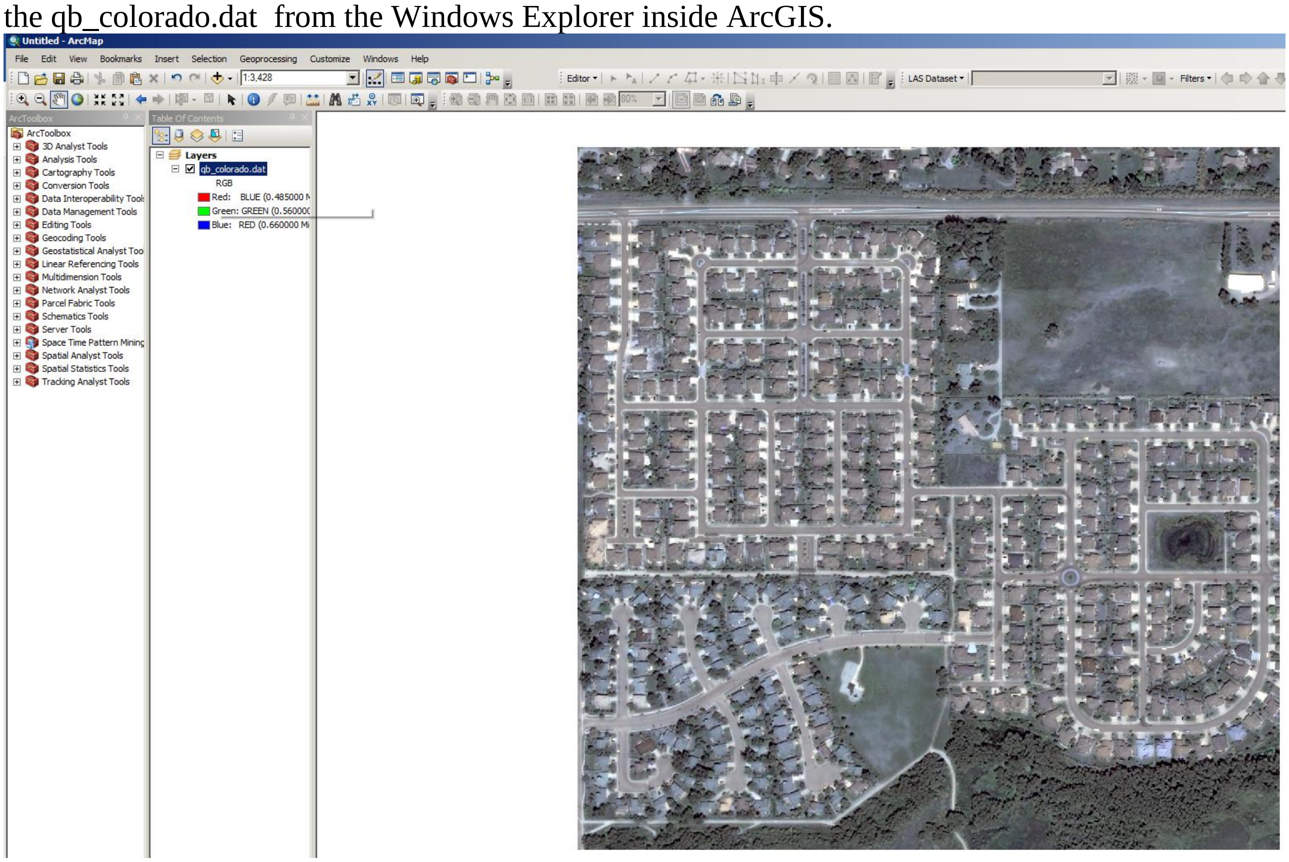
Q20 - What does spatial resolution means in respect with reality, or with the ground, to be more specific?

Q21- North is straight up in this image. Given this information, what is the position of the sun and what effect does this sun position have on the image?

Q22- Move to a portion of the image covering water. Some areas of the sea are black; others have patterns and are lighter. Provide some possible explanations for this variation in the water.

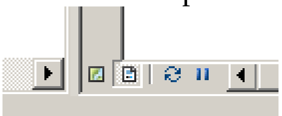
2.5.2 Labeling

ArcGIS is a platform for spatial data analyses. Therefore, most of the work is done within ArcGIS environment. However, once the analysis is completed, usually maps and other products are produced. Each map must contain at least three elements: the north direction, the scale bar, and a legend. In many instances the besides these three elements, maps contain labels that enhance the map. In ArcGIS, there are three methods of labeling a map: by adding text boxes, by turning on the labeling option associated with each layer, and to use annotations. We will explore only two of them in this lab, as the native labeling option available under the Properties window is meaningful only for vector data, not for raster. We will use all three of them using the qb_colorado image, which is an image acquired by the Quickbird satellite of a portion of Colorado state. that is store in an ENVI format, meaning there is a .dat file and a .hdr file. The ENVI header file contains metadata for ENVI-format images. ENVI creates a new header file whenever you save an image to ENVI raster format. The header file uses the same name as the image file, with the file extension .hdr. To load the image, open the ArcGIS and drag and drop the qb_colorado.dat from the Windows Explorer inside ArcGIS.

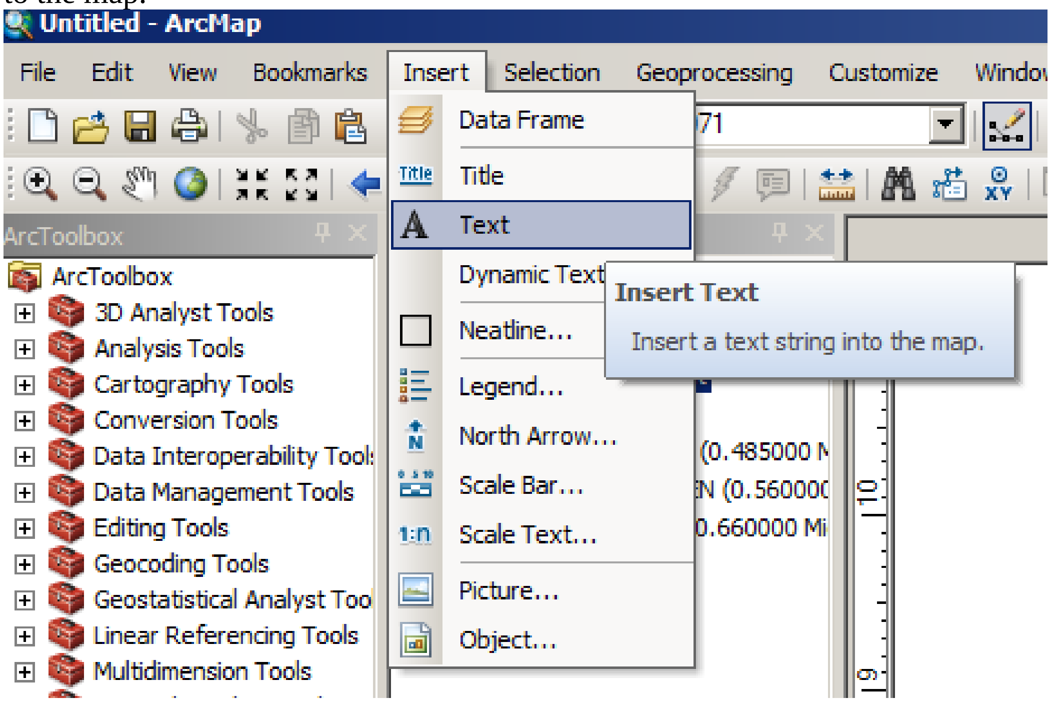


2.5.3 Text boxes

The natural way to add a label to a feature is by adding text boxes over the area that you want to label. To execute this task, you have to switch from the Data View to the Layout View first. The button that implements this task is located in the lower-left corner of the image viewer.

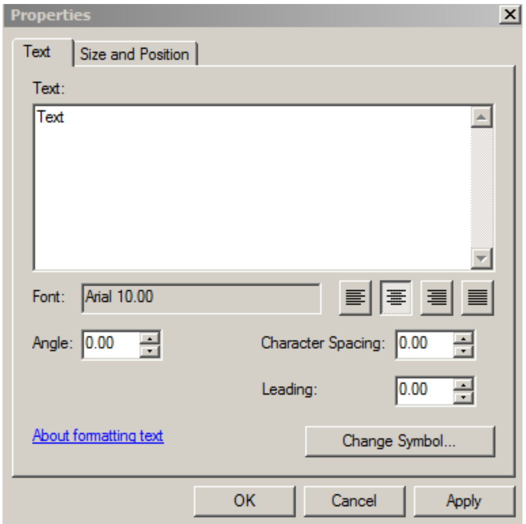


Within the Layout Viewer, the Insert option from the Menu bar offers the possibility to add Text to the map.



Selecting Text will add a text box in the middle of the screen, but with limited visibility.

To ensure that the label is position properly and visible, you should drag it over the feature that you would like to label. Let say that you want to label the filed in the right upper corner as “Central Park”. This means that you want to change the default text within the box, which is “Text” with “Central Park”. To execute this task with right-click on top of the text box or double click the box, which will open the Properties window.



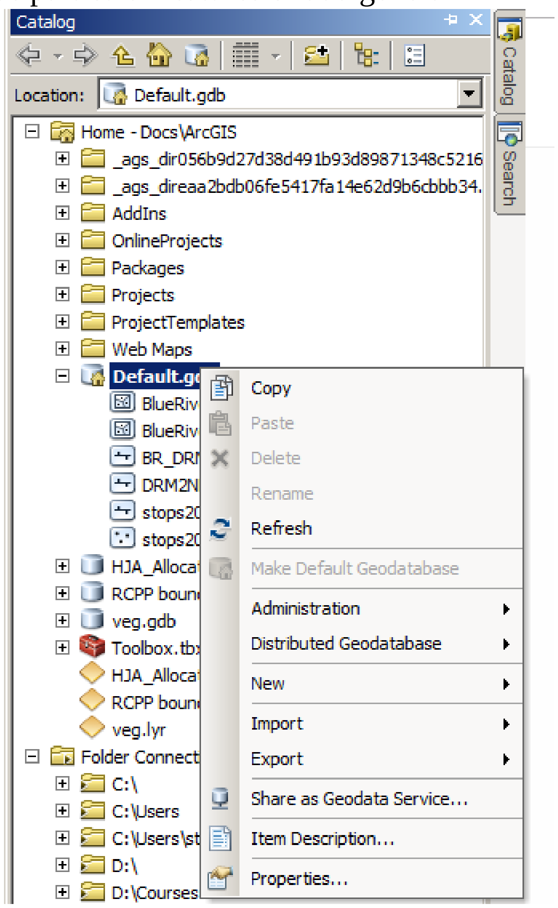
In this window, you will replace the default text with Central Park, change the font type from Arial to bold Courier New of size 25, and the font color from black to red. Also, rotate the box with 30 degrees, such that will fit better on the diagonal of the park. All these options are accessible by clicking the Change Symbol… button. Once finished, click Apply button to see the changes. If you like them, Press OK. To position the label on top of feature, drag the textbox on top of the feature.

Q23. Include a snapshot of the final labeling. It should look like the image below:

NOTE: The text box option is not recommended, except in limited situations, as the box location is NOT synchronized with the image. This means that if you pan the image, the box will not move with the image.

2.5.4 Annotations

Using annotation is another option, the preferred option, in ArcGIS for storing Text to place a raster that is a part of a map. Annotation can be used to describe particular features or add general information to the map. With annotation, the position, text string, and display properties are all stored together and are all individually editable. Annotation provides flexibility in the appearance and placement of your Text because you can select individual pieces of Text and edit them.

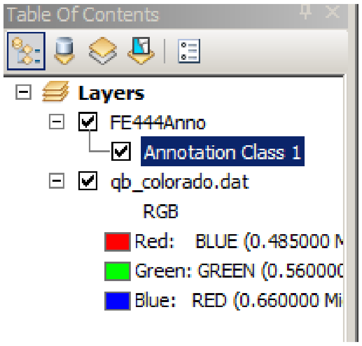


However, annotations for raster are available only thru geodatabase—that is, an annotation feature class stored in a geodatabase. ArcMap can be used to create and edit annotation features.
To create an Annotation feature first, you simply right click on the geodatabase that will store the Annotations, in this example, the Default.gdb, then select the New… option.

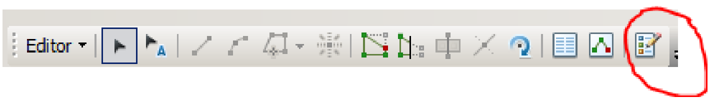
This will open a new window from which you should choose the Feature Class. The new windows triggered by your selection will lead you to the creation of an empty annotation layer. From the Type drop-down menu, select Annotation Feature and give it a name, such as FE444Anno (no space in the name).

Click Next when finished, which will open the coordinate system window. Because you would like to have related to the raster, you should choose the Layer option from the three available options, then select the WGS_1984_UTM_zone13N, which should be the only available option there (if more layers are added to the map, then more coordinates system could be there). Click Next when done, and choose the default XY Tolerance of 1 mm.

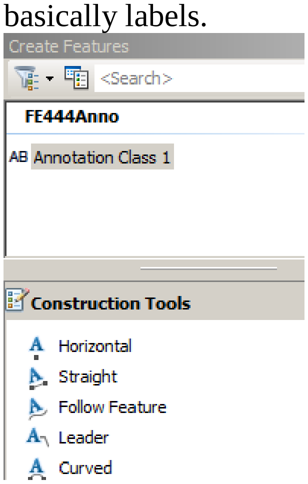
The Next press will trigger the reference scale for the annotations, which is usually defined by your project and size of the area covered by the image. Let's choose in this case 1:24000. For the next option, choose the default values. Once completed, a new layer will be added to the map, called FE444Anno.



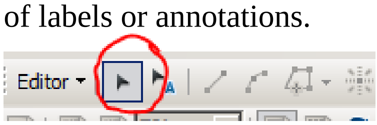
To start adding labels to the image the editing option would be used, which basically will identify a label with features with the FE444Anno layer. This is executed similarly to any editing icon in ArcGIS. First, you turn on the edits by choosing the Start Editing from the Editor toolbar. Then choose the Create Features button:



This will open a new window on the right of the map that controls the creation of new features, basically labels.



Now you are able to create your first label, which again will start with the word Text. Delete it, and type Central Park. When completed, place the label in the area that you consider appropriate. If you want to rotate it do so, but you can alter it later on the angle. After you place your first label, you can click the arrow within the editor toolbar, which will stop creation, for the moment, of labels or annotations.



Now select the label that you created and change the color and font to red and Times New Roman, respectively. This option is available thru the Attribute tab, located at the bottom of the Attribute Editing window. If you want to change the angle you can to also this. Once done, press Apply. You started from something like the left image and ended with something like the right image.

Once done with adding labels (annotations), press Stop Editing, then save your edits.

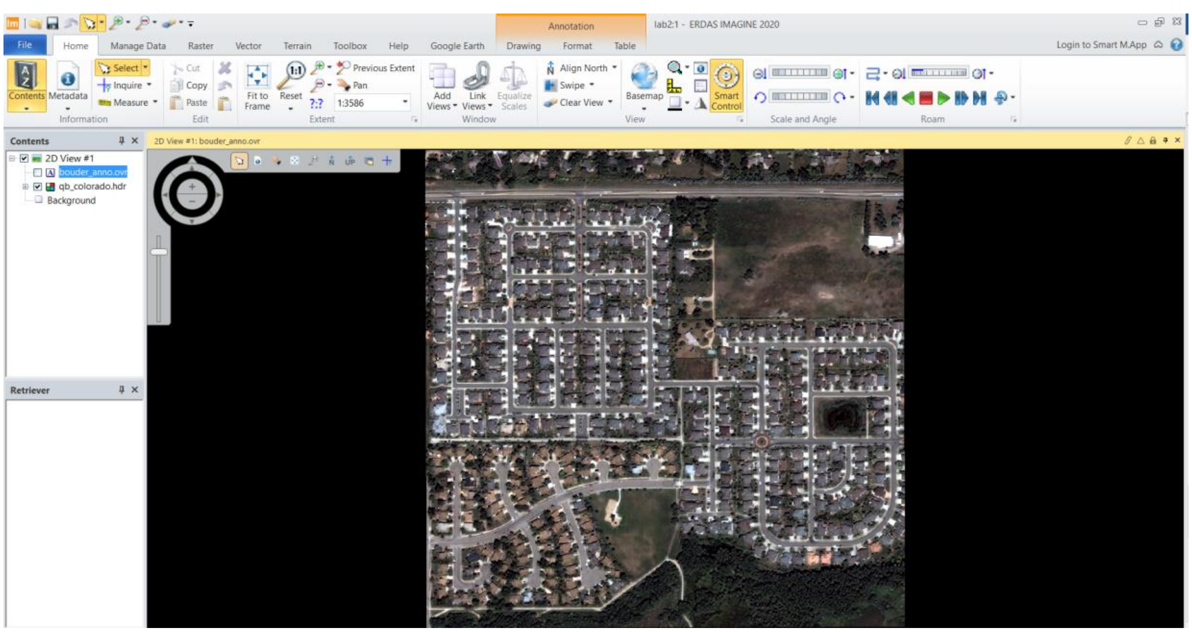
NOTE: Panning or zooming the image will preserve the relative location of the lalbels (also move the labels), which is what you want.

Q24. Create a new personal geodatabase, FE444, inside which you will place you annotations.
Q25. Include a snalshot of your annotation.

2.6 Advanced topics: ERDAS

2.6.1 Load an image

Load the panchromatic QUICKBIRD (Quickbird.tif) image of Vancouver. To load this image, in the main menu bar (above) go to File/Open, and then navigate to the place on your computer where you have the files associated with this lab. Once selected, the image will then appear in the main window and the layer details will show in *Layer Manager* area. Also, you can go to File/Open/Data Manager, and then navigate to the place on your computer where you have the files associated with this lab. If you want to move the image, click  on the above tool bar. **T**he display image shows below.



Q26. Include one snapshot with the window showing the coordinate system and the datum.

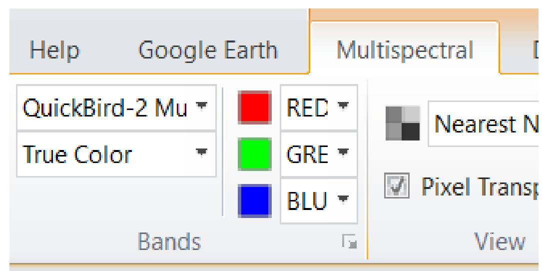
2.6.2 Annotations

Annotation is typically Text, but it can also include graphic shapes—for example, boxes or arrows—that require other types of symbology. Annotations are stored in the annotation layer. All features in an annotation feature class have a geographic location and attributes.

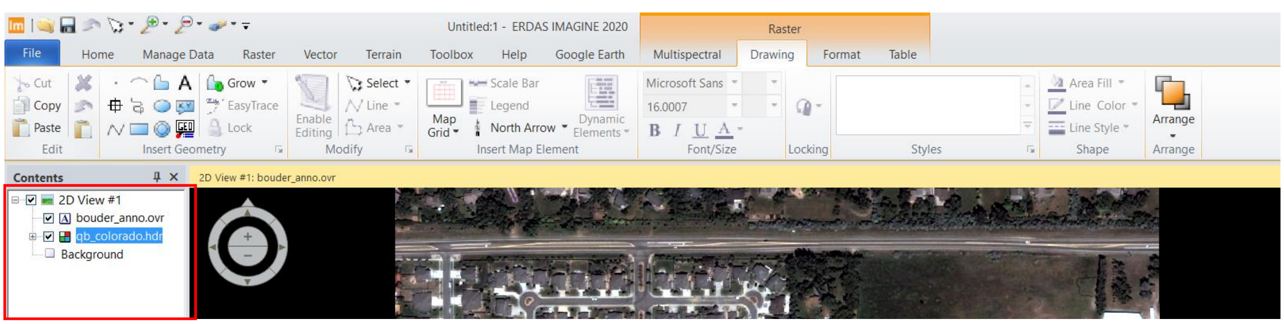
2.6.2.1 Creating an Annotation Layer

In the steps below, you will create one annotation layer. Annotation layers and vector layers are different in that you can have multiple annotation item types within an annotation layer. A single annotation layer may contain a combination of text, polygon, symbol, and other annotation items.

1. Load the QUICKBIRD image of Boulder, CO (qb_colorado.hdr) to the main View.
2. Once you added the image to the main View, **Raster | Multispectral** tab will activate.
3. The **Bands group under the Multispectral tab** allows you to use the spectral reflectance of the selected image. If the sensor is known, it will be identified in the **Sensor Type. However, if you can not see the correct Sensor Type for** qb_colorado.hdr, you can choose **QucikBird-2** as the **Sensor type**. You can change the band combination by using the drop-down menu from the **Band combination**. Select **True Color**, so the image would be displayed as it was seen by human eyes.



4. From the menu bar, select **File > New > Annotation Layer (under 2D View #1 New Options).** Then **Annotation Layer** dialog appears.
5. Enter "**Boulder_Anno"** as the File Name.
6. Click **OK**. ERDAS adds the new annotation layer to the **Content pane.**
7. Once you click on the **"boulder_anno.ovr"** file in the **Contents pane,** the **Annotation Tab** will enable.



NOTE: When you are performing Annotations, check whether the annotation layer (boulder_anno.ovr) is active. If not, select it and start working on annotations.
Always keep the annotation layer on top of the content pane list. If not, you can drag it and place it on the top of the list.

2.6.2.2 Adding and Saving Annotation Items

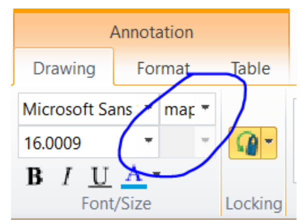
In the steps below, you will add several types of annotation items. When you created the annotation layer, the Text Annotation tool was automatically enabled.

1. In the Image window view, find a park area in the scene to label.

2. Navigate to **Annotation | Drawing > Insert Geometry** group. From that group, select **Text** icon A. Now you can see that the cursor has changed into **I** shape.

3. Place your cursor in the middle of the park and type **Central Park.** The annotation item is added to the Annotation Layer. Now click the cursor somewhere in the image, and you will see that the cursor symbol has changed to an arrow mark again.

4. Now you can click on the text, and it may look like this . Also, you can see the features in the **Annotation| Drawing** tab are active now. Change the font type and size appropriately so that you can see the text.



NOTE: You can use the **Fron/Size** group under Drawing Tab to specify how you want text to appear. If the annotation is georeferenced, then you can specify the size, position, and other element properties in either map or paper units. This works in conjunction with **Font Units** option. Your selection not only affects the numbers that you use, but the way that your elements may be displayed in a View, Map View, or on paper when printed.

Map: The annotation size is related to the geographical coordinates.

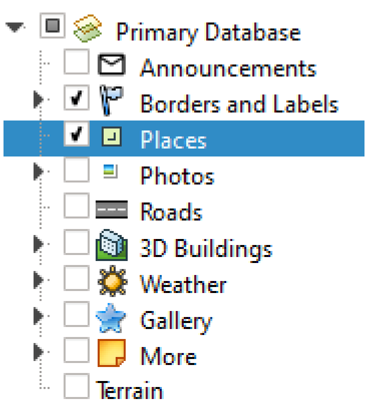
Paper: The annotation size is related to units of the View or related to paper size.

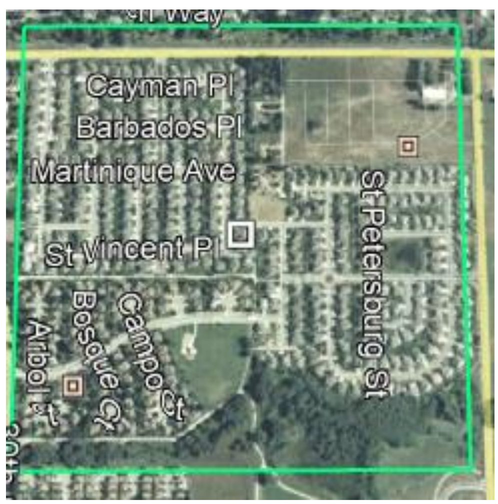
(**source:** ERDAS Image Help)

5. Select the "Central Park" text box again and go to **Annotation | Format.** Form **Styles** group, you can select an appropriate style for the text box. Also, you can change the **Area fill** and **Line-color/style as well.**

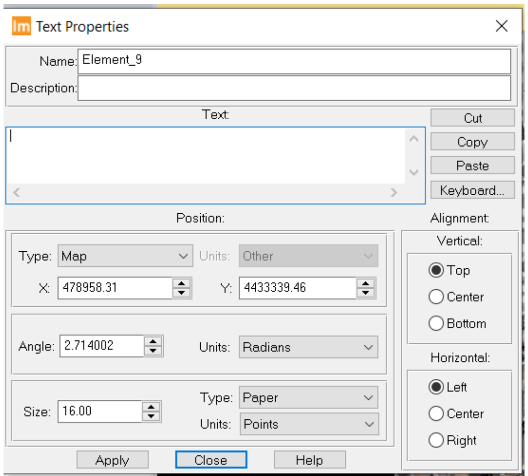
6. Use the **Pan tool** 

 to find a different park in the scene.
7. Now select the “qb_colorado.hdr” image from the **Content pane**. Navigate to **Google Earth Tab and Connect to Google Earth.**
8. Select **Match GE to View** and **Export View Footprint**. A separate Google Earth Window will open and zoom in to the image area. Select the Places Checkbox, so you can see the names of places.





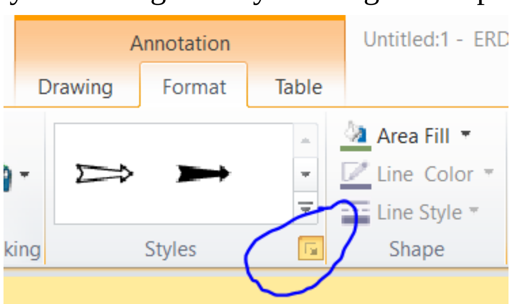
9. Now you can identify another park located in the lower part of the “qb_colorado.hdr” image with the help of Google Earth.
10. Redo steps 2 and 3, name the park. You may have to select the annotation layer gain to perform this task.
11. Change the **Font Type** to **Times New Roman**, the **Font Size** to 14, and change the font color to red.
12. Select and double click on the text box. The **Text Properties** dialog box will open.

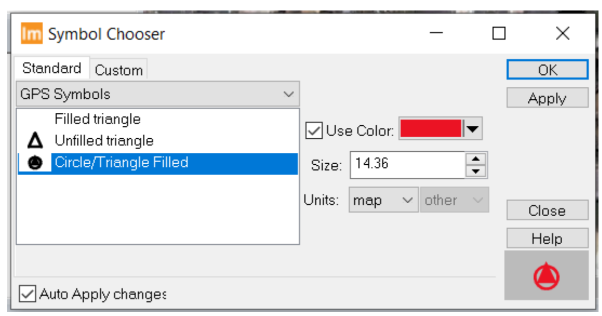


13. Set the **Angle** to 20 degrees and set the **Vertical and Horizontal** positions as **Center.**
14. Using the previous settings, label two more items as text annotations.
15. Save the annotation layer before you continue—Right-click on `Boulder_Anno` in the Content Pane and select **Save Layer**.

Next, you will add symbol annotations to the scene.

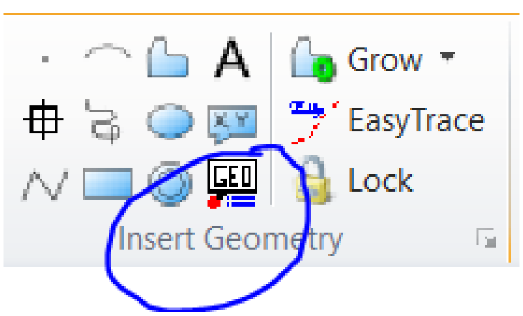
1. Click the **Annotation Layer** again and navigate to **Annotation | Drawing** and select **Point** icon . Now you will see that the **Styles group** under the **Format** tab is activated.
2. Click on the small arrow button in Styles Group .
3. Symbol chooser dialog box will open. From there, you can select different types of symbol categories by clicking the drop-down menu.





4. Scroll down and select **GPS symbols**, then select **Circle/Triangle Filled**. Change the color of the symbol to red using the color selection window.
5. Select **map** for units. Now you can navigate the cursor to the roundabout that is located between two parks and click. Now you can see that GPS symbol is added to the annotation layer.

6. Select **Place Geo point Properties** icon from **Insert Geometry** group under the **Drawing** tab. Next, the **Utility** tab will open. Select **Place GeoPoint** tab in **Utility | GeoPoint** tab. Navigate to the roundabout and click the top of it.

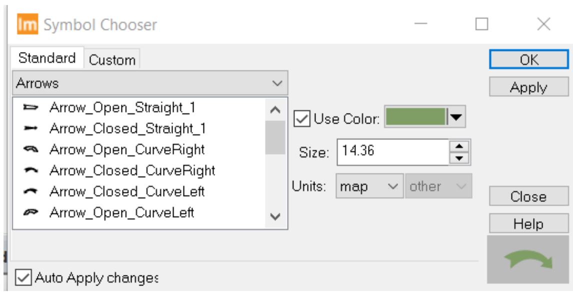


7. You will see that **GPS location** of the roundabout has been added to the annotation layer.
8. Add two other GPS symbols to the annotation layer.

Now add some arrow annotations.

1. Click the **Annotation Layer** again and navigate to **Annotation | Drawing** and select **Point** icon . Now you will see that **Styles** group under **Format** tab is activated.
2. Use **Symbol chooser dialog** box to select **Arrows** category. From there, you can select different types of arrows by clicking the drop-down menu.

3. Once the Symbol chooser dialog box opened you can select different types of arrow symbol categories by clicking the drop-down menu. Also, you can change the color of the symbol using the color selection window.



4. You can select **map** for units. Click Apply and check how well fits your size and color for the arrow symbol. Make any appropriate changes if it is necessary and click OK.

Next, add rectangles and resize them.

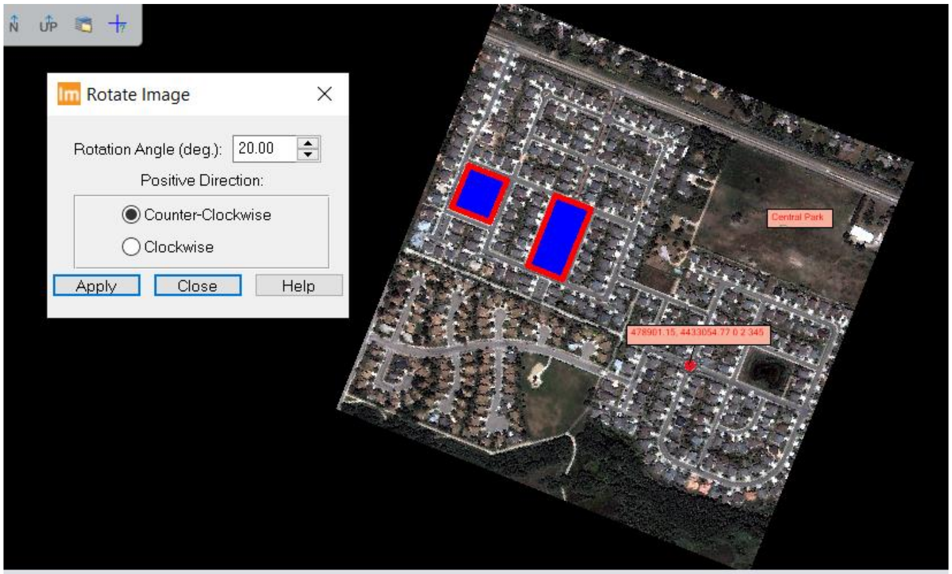
1. Click the **Annotation Layer** again and navigate to **Annotation | Drawing** and select point icon . Now you will see that **Styles** group under **Format** tab is activated.
2. Draw a rectangle that covers a block of buildings. Click to of the rectangle and **set the line style to 4 pt and color to red**. Change the area fill to **blue color.**
3. You can copy and paste this rectangle by clicking **Ctrl+C and Ctrl+V**. Paste it to an adjacent set of a housing block.

4. Click the **Annotations** drop-down on the toolbar and select **Rectangle Annotation**.
5. Resize the rectangle you drew by clicking and dragging on the selection handles.

6. Save the layer—Right-click in the annotation layer on the Content pane.

In the final steps for annotations, you will rotate the image and the annotation items.

1. Click **Home tab > Scale and Angle group** > **User-Defined Angle**. Then **Rotate Image dialog** box will open. You can set the **Rotation Angle** to 20 degrees and select **Counter-Clockwise** as **Positive Direction**. Then click **Apply and Close.**



2. By default, polygon, polyline, rectangle, ellipse, and arrow annotations rotate with the underlying image. Text, picture, and symbol items do not rotate with the image.

3. You can save the layer by selecting Select **File > Save** > **Top Layer** to save the annotation layer. You can select **File > Save As** > **Session** to save the entire work.

NOTE: You can open the attribute table for the annotated layer by clicking the Annotation | Table.
This table will help you to see the details of each item you added to the annotation layer.

bouder_anno.ovr

Row	ID	Type	Name	Description	ULX	ULY	LRX	LRY
1	37	Text	Element_9		478927.79	4433382.43	479045.47	4433347.66
2	110	Symbol	Element_110		478891.66	4433403.68	478893.51	4433401.04
3	119	Symbol	Element_119		478642.24	4433074.99	478662.26	4433067.39
4	121	Symbol	Element_121		478958.59	4433345.52	478972.96	4433340.06
5	153	Symbol	Element_153		478896.84	4433062.92	478908.96	4433048.64
6	172	Group	GEOPOINT-GROUP		478780.27	4433126.39	479035.03	4433053.36
7	178	Rectangle	Element_178		478568.04	4433248.61	478644.76	4433106.68
8	189	Rectangle	Element_178		478410.08	4433238.75	478485.99	4433156.18
9	567	Text	Element_567		478958.55	4433383.65	478973.00	4433368.79
10	568	Symbol	Element_568		478911.41	4433392.00	478924.80	4433377.61

***Q27**. Identify two other areas in the image that could be a park/ housing scheme and label them as you see fit (you can use Google Earth for this), create the annotations, and include a snapshot with the two labels together with all the annotation items you have added so far.*
***Q28**. Include a snapshot of the Attribute table of annotations.*

3 Basic Image Analysis: Visual analysis and PLSS

3.1 Objectives:

- Loading and exploring different types of remotely sensed images
- Making colour composites
- Displaying individual spectral bands
- Viewing pixel grey values (digital numbers)
- Histograms
- Image Enhancements
- Working with PLSS maps
- Extract information from US Topo 7.5" maps

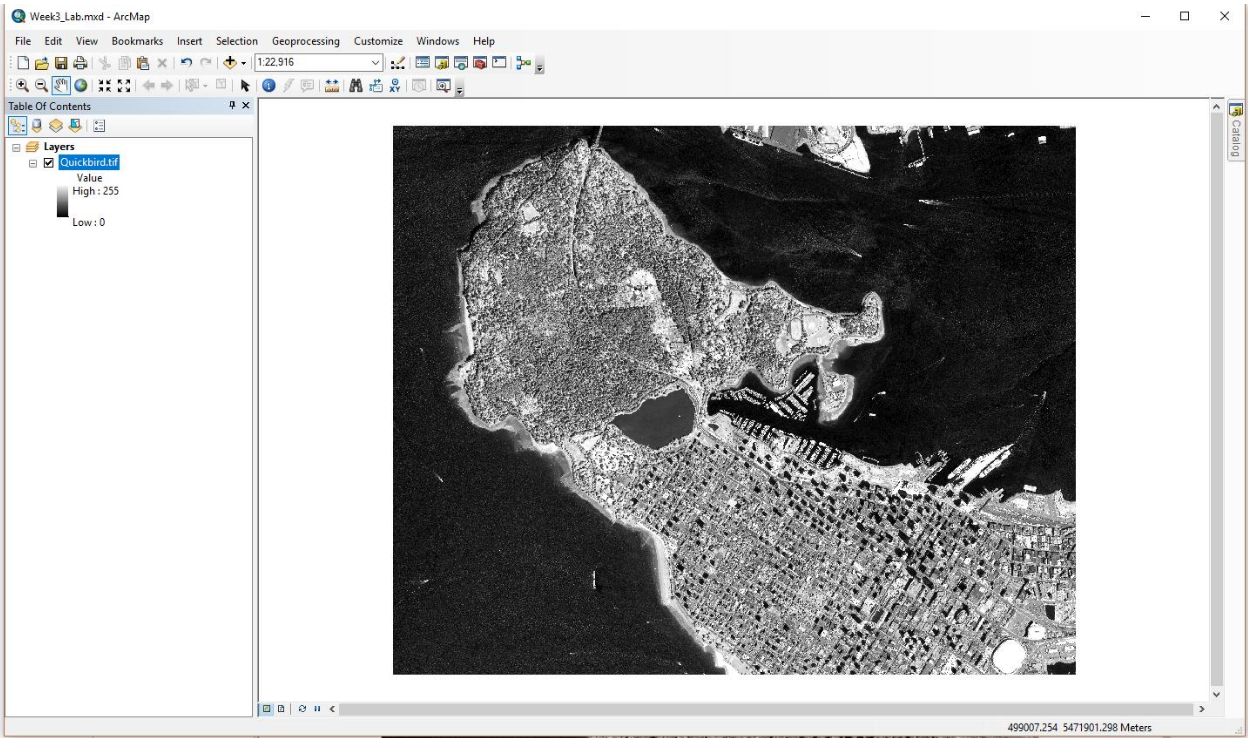
3.2 Description of the files

- Quickbird.tif is a QuickBird multispectral image of Vancouver BC, Canada
- Landsat7-pan.tif a panchromatic Landsat 7 ETM+ image of Vancouver
- Landsat99.tif is a Landsat 7 ETM+ of Vancouver BC, Canada from 1999
- Landsat85.tif is a Landsat 5 of Vancouver BC, Canada from 1985
- VanTM is a Landsat TM of Vancouver BC, Canada from 2006
- WestVan.tif is an image of Vancouver BC, Canada

3.3 Visual image analysis

3.3.1 Basic visual image analysis

The first image is a panchromatic QUICKBIRD (Quickbird.tif) image of Vancouver. To load this image, click on [icon] in the main menu bar and then navigate to folder where you saved the images for lab 3. If you cannot find the folder, then in the Add Data window, [icon] click and connect to the folder where the images for the week 3 were saved. Once selected, the display image should look like below.



The second image to load is the “Landsat7-pan.tif (**Figure 1**). Once two images are loaded, the images will be shown on the table of contents.

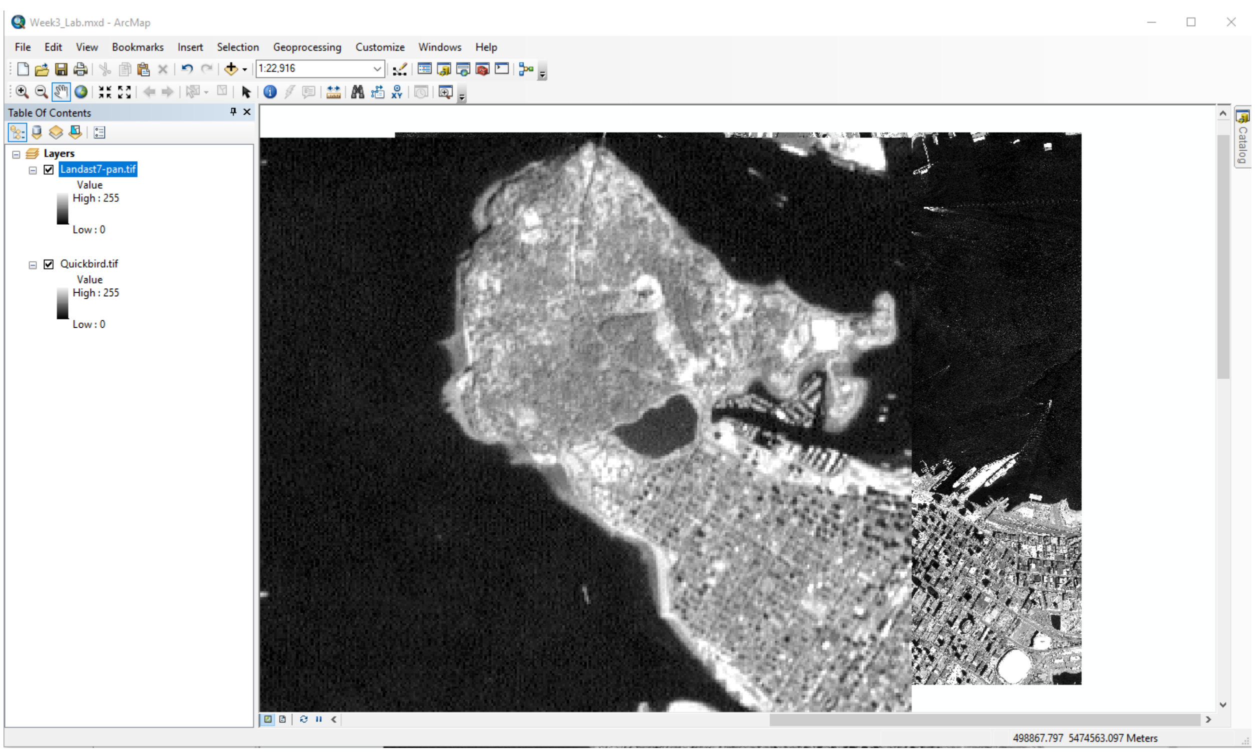


Figure 1. This is what should appear on screen when the first two images have been displayed.

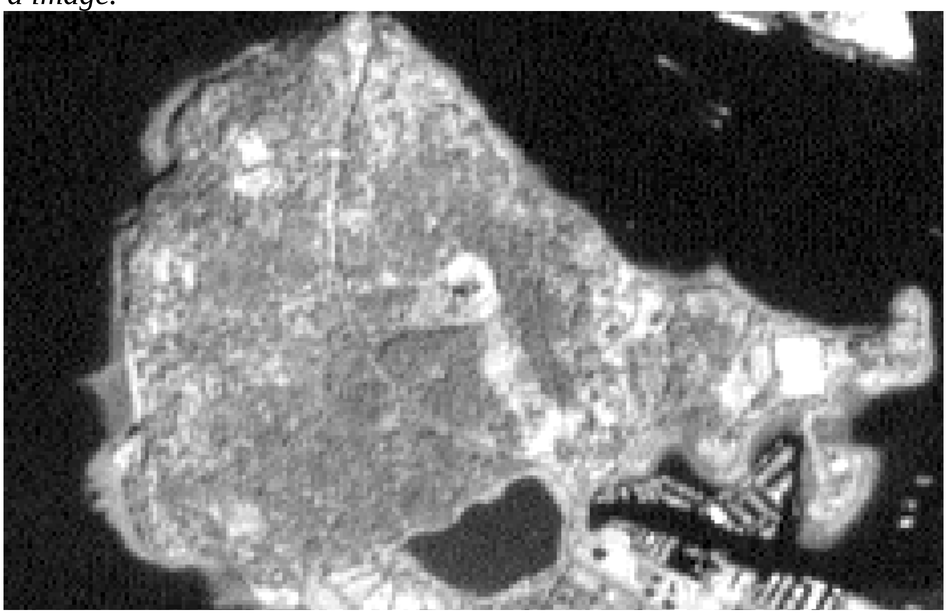
The second image is a panchromatic Landsat 7 ETM+ image of Vancouver. Briefly familiarize yourself with the characteristics of the Quickbird and Landsat ETM+ sensors., either by searching online or in the textbook.

To examine the information of the images, right-click on the image layer in the table of contents, then select **Properties**. Here you can find the image information by clicking on the **Source** tab. To manage the visibility of the image layers, in **Table of Contents**, check in the box next to the layer to display it and uncheck it to turn off the layer.

Q1*- What is the spatial resolution of the Quickbird and Landsat 7 images? For full credit, provide at least TWO ways of finding the spatial resolution.*

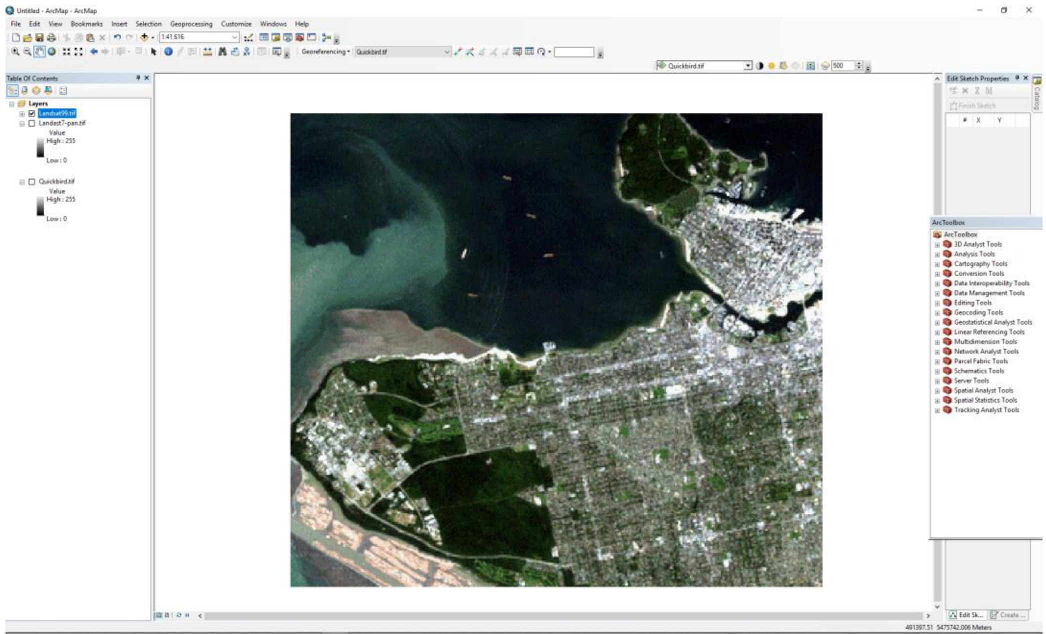
Q2. Why the spatial resolution of the panchromatic image is smaller than the spatial resolution of multispectral bands for Landsat image? Think on the reflectance – sensor interaction.

Q3*- Zoom over Stanley Park and list 4 differences you observe between the Landsat image and the Quickbird image.*



Q4*- List two possible uses/applications/advantages of using this Landsat scene over the Quickbird image.*

The third image we will look at is the “Landsat99.tif” image. Load this image in the same manner you loaded the last one. This time, you can see the image is colored, which is different than the other two. This is a “true-color” composite image taken at the same time as the panchromatic image that we just looked at. Remember what true-color means in relation to the last lab.



Q5- *Based on what you see in the images, how can you tell this true-color composite is from the same exact time as the panchromatic image?*

Q6- *What does true-color mean? Think on the relationship between the recorded wavelength and the displayed wavelength.*

Q7- *What is the spatial resolution of the Landsat99 image?*

The fourth image to open is the “Landsat85.tif”, which is also a true-color image from 1985. This means it must be displayed in the same manner as RGB color. This image has roughly the same spatial extent as the Landsat99 image.

Q8- *Zoom onto the UBC campus and list two obvious differences between 1985 and 1999 based on image comparison. What could cause the changes?*

Q9- *Why is the water around land from the Landsat85 image shown differently than water farther from the coast?*

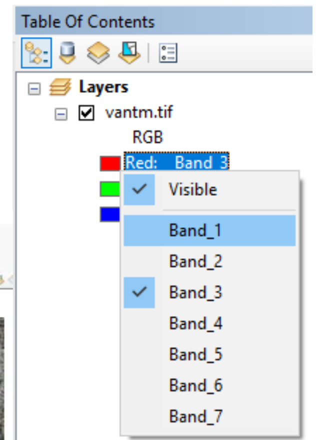
Next, you will load the “WestVan.tif” Image. This is an image with a spatial resolution of 60 m. This image has a coarse (comparatively) spatial resolution for a reason. This reason relates to the part of the spectrum that it is representing. To answer the next question, think back to your first lecture (in relation to wavelengths and frequencies) and the last lab (in relation to the characteristics of a certain sensor that were provided).

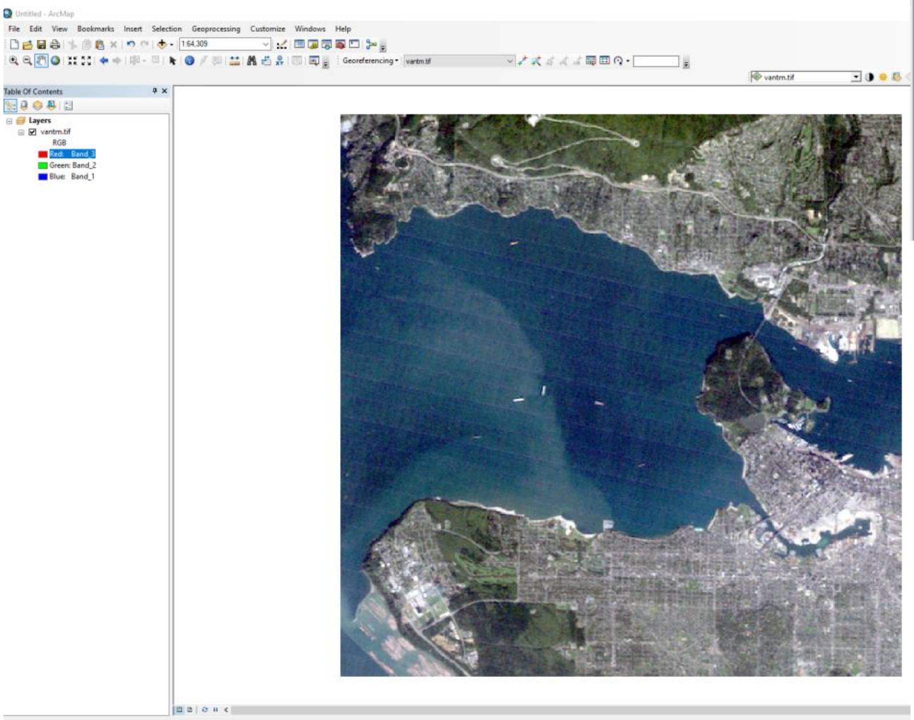
Q10- *What part of the spectrum is this image representing and why does this mean its spatial resolution has to be 60 m (coarser than all of the other spatial resolutions we’ve been dealing with so far in this lab)?*

***Q11**- This image is from the early morning in summer time. As a result, the urban areas are whiter than the forests and the water. Why?*

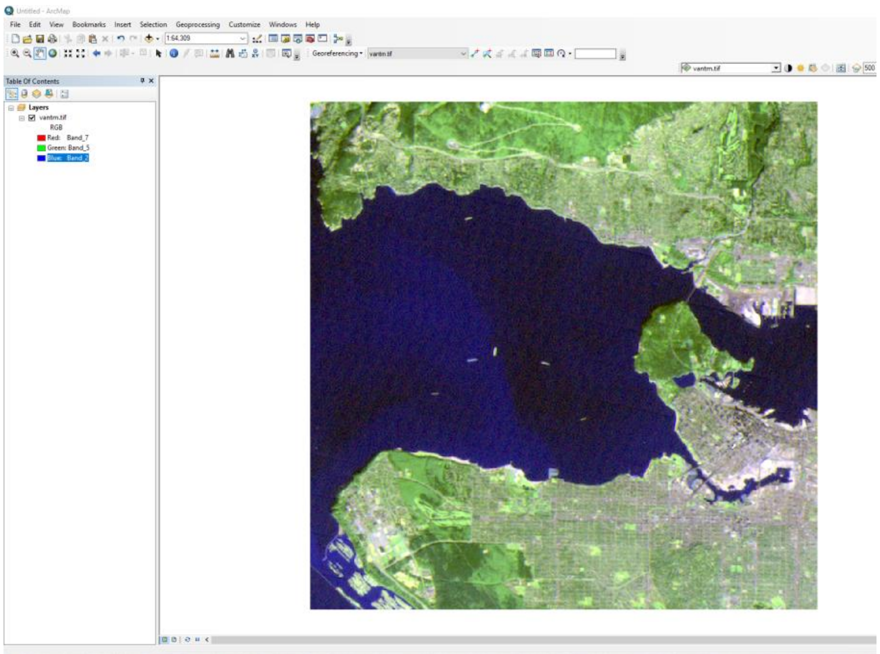
Remove all the images that have been opened by right-clicking the image layer, and choose **Remove**. The next image we will look at, vantm, is a Landsat Thematic Mapper (TM) image of Vancouver from 1984. The image consists of 512 lines and 512 cell (columns), forming a grid of 512 x 512 pixels. The image file has seven bands, identified as bands 1 through band 7.

1. Add the image *vantm.tif* to ArcMap.
2. In **Table of Contents**, click on band Red, set Red to band 3; similarly, set Green to band 2, and Blue to band 1. And the image will be shown as below.





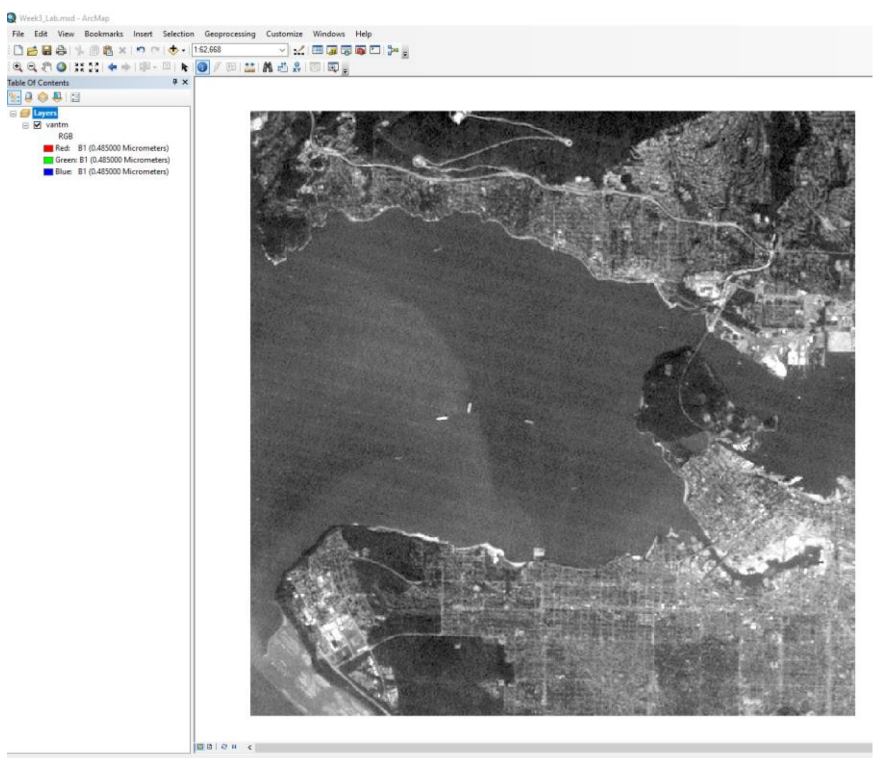
To create false-color composite, just repeat the steps above but assign Red: 7, Green: 5, Blue: 2.



Q12- *What makes this a false-color composite? What bands are represented by what colors? What parts of the spectrum do these bands represent?*

You can experiment with different band combinations by loading different bands as Red, Green, and Blue. A standard false-color composite, for instance, has Band 4 on Red, Band 3 on Green, and Band 2 on Blue.

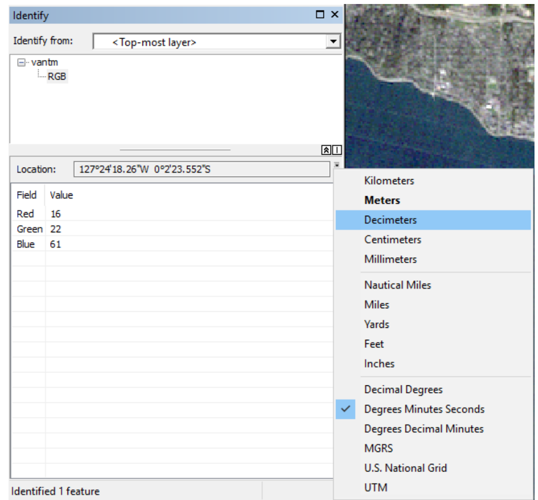
To display band 1 as greyschale image, assign Red: 1, Green: 1, Blue: 1 by using the same step as before.



3.3.2 Viewing image coordinates and grey values (digital numbers)

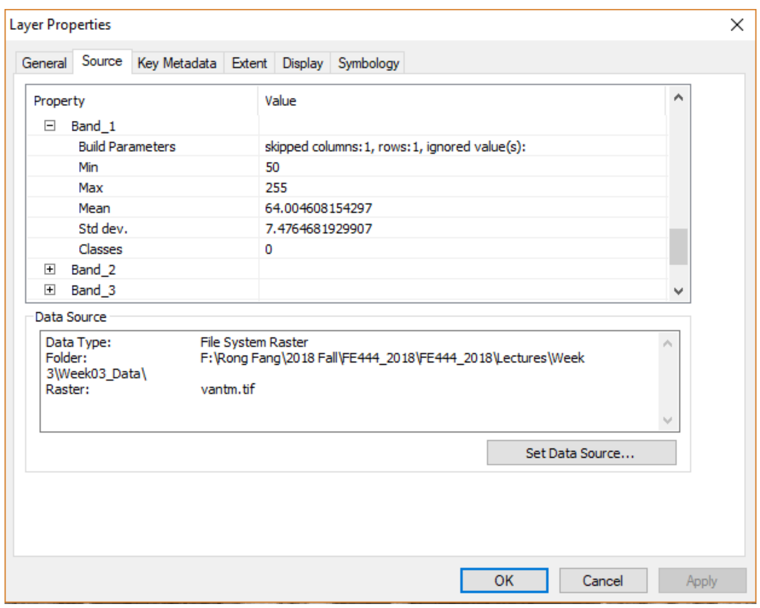
For most remote sensing based applications, you need to know the image coordinates. The coordinates of any pixel can be displayed using the *Cursor Value* tool. Similarly, the value of digital numbers at any given point (based on cursor position) is also displayed.

1. Select **identify**  from the tollbar
2. Move your cursor over the image and examine the changes in DN values and coordinates by clicking on pixel of interest.
3. To change the coordinate units, click on the dropdown list in Location, then you can choose the interested units.



3.3.3 Statistics

Summary statistics for DN of each band can be examined in the **Properties** of the data layer. Right-click on the image layer in **Table of Contents**. Under the **Source** tab of **Layer Properties** dialog box, scroll down, you will see the band statistics.



3.3.4 Image enhancements

The pixels in a data file that make up an image can have any value: negative, positive, integer, or floating point. When the image data are visualized on screen, they are displayed as brightness values for each screen pixel. A data pixel with a larger value will be brighter than one with a smaller value. However, unlike the image data, screen pixels can only have 256 unique brightness values, varying as integers between 0 and 255 (if the screen display is 8 bit). Clearly, this limitation prevents the data from being displayed with brightness exactly equal to their real value. For example, how would you display a negative data pixel or floating point data ranging between zero and one?

Stretching the image data refers to the method by which the data pixels are rescaled from their original values into a range that the monitor can display — namely, into integer values between 0 and 255. For example, if the image data were floating point values that ranged between -1.0 and 1.0, the image might be stretched such that data value of -1.0 is assigned a brightness of 0, and data with values of 1.0 is assigned a brightness of 255. All of the intermediate data values would have new stretched values based on a simple model or *stretch type*. Commonly, a linear stretch type is used so that the stretched data values maintain the same relationship to each other as the original data (e.g., the relative distance between two stretched values is the same as the relative distance between the two original data values). Other stretch types use different models to assign the intermediate values, such as gaussian, equalization, or square root functions.

1. Display the first band of the *vantm.dat* image.
2. Open *Identify* by click
3. Roam around in the display window and try to find some of the brightest and darkest pixels in the image (you can zoom in or out to use the resolution you think it is appropriate for you to operate).
4. Using the *Identify* widget, note the data value, the corresponding screen value (i.e., the stretched value), and the image coordinates for one of the bright and dark pixels.

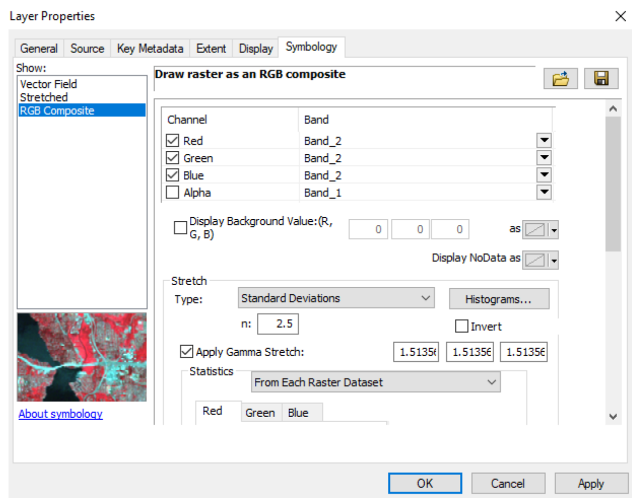
Q13*- bright pixel*: coords: ________________data value: ______
Q14*- dark pixel*: coords: _________________data value: ______

The exact method that is used to rescale the image data into brightness values can make a drastic difference in the way that the image looks. Thus, it is quite common to adjust the parameters of the stretch in order to maximize the information content of the display for the features in which you are most interested. This process is referred to as *contrast stretching* because it changes contrast in the image -- the relative differences in the brightness of the data values (i.e., increasing contrast means that the dark pixels are darker, and the bright pixels are brighter, so the brightness difference between the two is increased).

For example, consider an image whose data numbers (DN) are integers that range between 35 and 85 (only 51 different data values). If this image was stretched with a simple “one-to-one” model where a data value of 0 is assigned 0 brightness, and a data value of 255 is assigned 255 brightness, then the image display will be quite dim (since the brightest pixel is only a brightness of 85). This stretch produces a low contrast image because a difference in data value of 1 unit is represented by a difference in brightness of 1 unit. Furthermore, much of the range of available screen brightnesses is not being used because there are only 51 different values in the image data (i.e., there are no pixels with a brightness between 51 and 255, so these brightnesses are unused). The image contrast could be maximized by assigning a brightness of 0 to the minimum data value of 35, a brightness of 255 to the maximum data value of 85, and linearly stretching the remaining 49 data values through the rest of the available brightness range. This increases the contrast because adjacent data values now differ by several units of brightness rather than just 1, making it easier to visually distinguish slight differences in the data values.

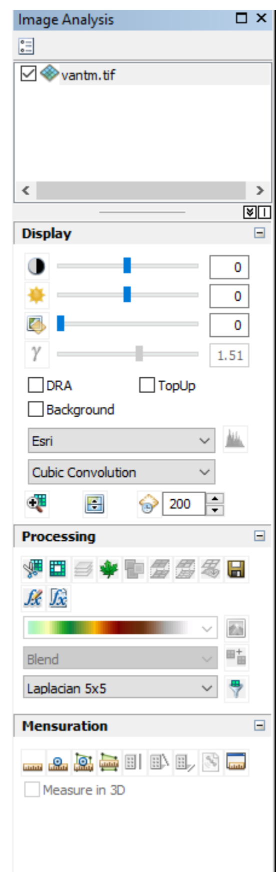
5. There are several ways in ArcGIS you can use to stretch the images:

1) Right-click on the image layer → **Properties**, choose **Symbology**, and you can specify the stretch type. Esri is the default stretch type of ArcMap. Other stretch types are standard deviation, histogram equalize, minimum-maximum, histogram specification, and percent clip.



For the details of each stench type, you browse the ESRI website.

2) You also use **Image Analysis** to change the stretch type. In the main menu, choose **Windows** → **Image Analysis.** In the Display panel, you can choose the image stretch type.



Q15*- Display the image with Percent Clip stretch. What features does this Percent Clip stretch highlight?*

Find an area in the image with the very brightest pixels that are nearly saturated with white pixels). Zoom in on this white patch so that the zoom window contains mostly saturated white pixels. Using the *Identify*, note that even though the pixels in the Zoom window all appear to have the same brightness, they actually do have slightly different data values.

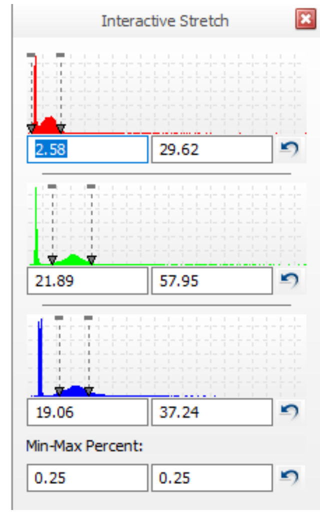
Contrast stretching an image for display does not affect the original image data. It is important to remember that whenever numerical processing is performed; the original data from the file are used, not the contrast stretched data that are displayed.

6. Add *vantm.dat* again to ArcMap, and display it as a grayscale image. The two images will probably look quite different, but remember the data in each view are actually identical; it's only the stretch that is different. Using the notes you wrote down in step 4, Use *"Go To XY"* in the main menu to get the cursor value for the new stretching image. Next, record the value of the same pixel from the vantm image that has the PercentClip stretch.

Q16*- What data value is reported by the Cursor Location/Value?*

Both images should report the same data value for the same image coordinate, regardless of the stretch applied.

3.3.5 Custom stretching



In this exercise, you will learn how to use ArcGIS's tools for complete manual control of the image display's contrast stretch.

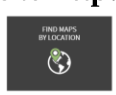
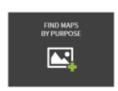
1. Set the image (*vantm.dat)* to display as Red: 7, Green: 5, and Blue: 4.
2. Open **Image analysis**, click **Interactive Stretch**  .
3. Adjust the stretching range of each band; you will see the change of image effect.

3.4 PLSS and Map reading

3.4.1 Overview

For foresters and other resource professionals, map reading, and the ability to make calculations from maps is an invaluable skill. This lab will present you with several general problems of distance and bearings and other map-reading information needs. You will be working with a digital map, and for several of the problems, you will need to zoom in to the area and then print out a hard copy to make your measurements from.

Step 1 (*NOTE: USGS changes their website frequently, the steps to obtain the PDF may be different by the time you are doing this lab*)

a. Open an internet browser and navigate to **http://store.usgs.gov**
b. Press the Find Map By Location box,  , then select US Map locator 
c. A new window will appear and in the Map Locator search bar, type in "Marys Peak OR. If the search does not work, then scroll down to Find out more about the following maps and select the US Topo Quadrangles icon:

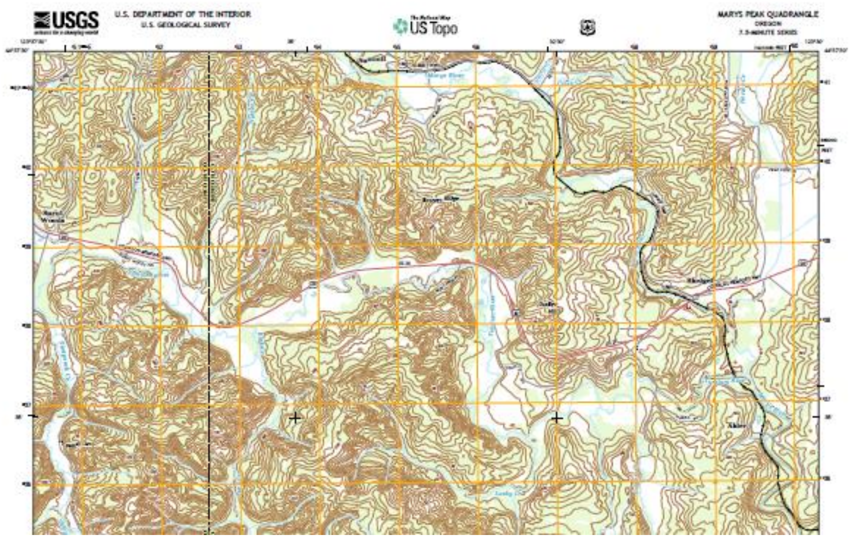
d. In the new interface, choose the country USA, and type in the keyword box "Marys Peak". If more products are available choose "MARYS PEAK, OR" with the following attributes:

  - MARYS PEAK, OR TNM GEOSPATIAL PDF 7.5X7.5 GRID 24000-SCALE TM 2016Survey date: 2020
  - Scale: 1:24,000
  - Format: Flat

e. Press on View/Download Free PDF of the 2020 USGS US Topo 7.5 minutes

f. This should start downloading a pdf file that contains the 7.5 minute map. If the PDF opens in your browser, click the ‘download’ button on the PDF, next to the print icon.

Step 2. Open the map in Adobe Acrobat or whatever PDF viewer you use



a. For each question or set of questions, you may want to zoom in on the area and print that section out rather than work from the screen

Note that the sections are generally standard 1 mile sections, in other words, each side is 5280 feet long in ground distance units. When you print out an enlarged section, you can use this as a ratio to get other ground distance lengths.
For example, if the side of a section is 3.5” in length and you have a distance of some other measure that is 1.8” in length:

$$\frac{5{,}280\ ft}{3.5"} = \frac{x}{1.8"}$$

The new distance in ground units is $\frac{(1.8")x5{,}280\ ft}{3.5"} = 2715.4\ ft$

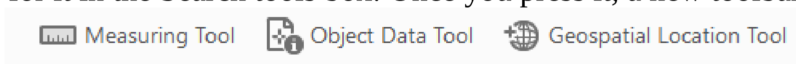
3.4.2 **Exercises**

3.4.2.1 Distances

Locate the road end in the northeast ¼ of section 16, T12S, R7W. There is a small hilltop with an elevation of about 1800 feet. The watershed boundary runs through this point. If you were to follow approximately this watershed boundary and hike to the top of Mary’s Peak:

***Q17** - How far would you have to hike? In feet and in miles?*

***Q18** - What would be the average percent grade of the hike?*

To answer Q17 and Q18 you should use the measuring feature option from Adobe Acrobat Reader that lets you measure objects on the map. The tool is located in many instances on the right side of the screen (depending on where you have the toolbar located):  . If you do not find it just search for it in the Search tools box. Once you press it, a new toolbar will appear on top of the image: 

Once you are selecting the Measuring tool, you have the option to measure distance, perimeters, and areas: 

3.4.2.2 Coordinates and Bearings

A helicopter has been ordered to do a water drop on a small fire burning on a saddle ridge in the NE ¼ of section 19, T11S, R7W. The fire center is near the peak with an elevation of 1120'. The nearest water source is a small pond in the SW ¼ of section 21 (the smaller of the two ponds shown on the map).

***Q19** - What are the UTM coordinates of the pond center?*

***Q20** - What are the straight-line distance and the compass bearing from the pond center to the saddle ridge area?*

3.4.2.3 Measurements of an area

***Q21** - You are interested in purchasing section 13, T12S, R8W. How many acres of non-forest land are approximately in the section?* Measure the square feet and then convert to acres. Remember that you can use the Measuring tool in Adobe.

3.4.3 Descriptions

Answer the following from the information on the map

***Q22** - What data were used to compile the map?*

***Q23** - What is the contour interval of the map?*

***Q24** - What are the horizontal and vertical datum of the map?*

***Q25** - If you needed more map information to the southeast of this map, which map would you order?*

Bonus question: What projective surface was used to create the map: cone, plane or cylinder? How do you know that?

3.5 Advanced Topics: Basic Image Analysis with ERDAS

Lab developed by Sudeera Galapita and Dr. Strimbu from Dr. Hilker, ENVI and ERDAS documentation.

3.5.1 Objectives

- Exploring different types of remotely sensed images
- Making colour composites
- Viewing digital numbers
- Histograms

3.5.2 Description of the files

- Quickbird.tif is a QuickBird multispectral image of Vancouver BC, Canada
- Landsat7-pan.tif a panchromatic Landsat 7 ETM+ image of Vancouver
- Landsat99.tif is a Landsat 7 ETM+ of Vancouver BC, Canada from 1999
- Landsat85.tif is a Landsat 5 of Vancouver BC, Canada from 1985
- VanTM is a Landsat TM of Vancouver BC, Canada from 2006
- WestVan.tif is an image of Vancouver BC, Canada

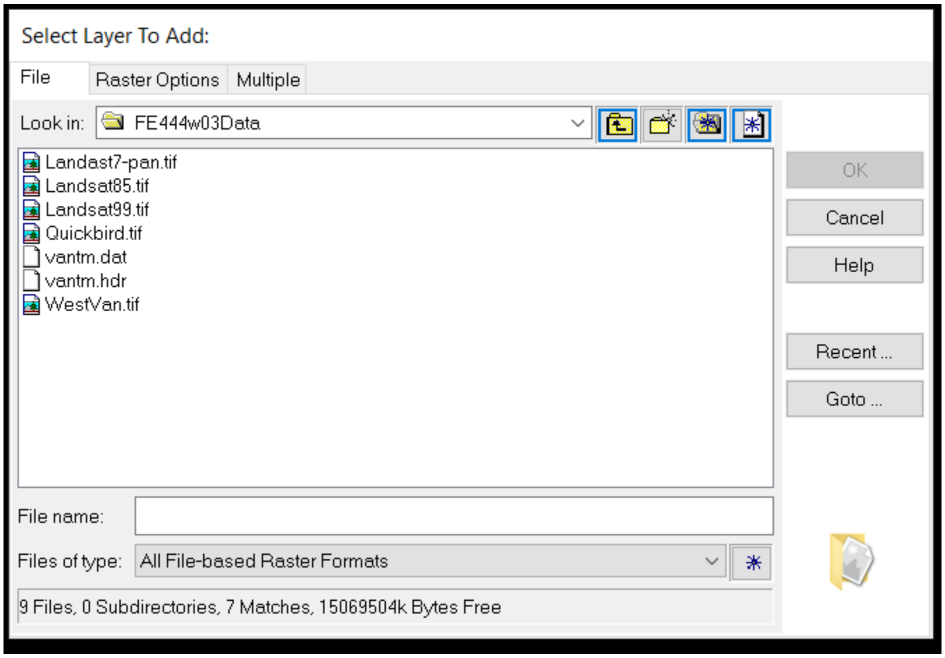
3.5.3 Exploring remotely sensed images

3.5.3.1 Load images

The first image is a panchromatic QUICKBIRD (Quickbird.tif) image of Vancouver.

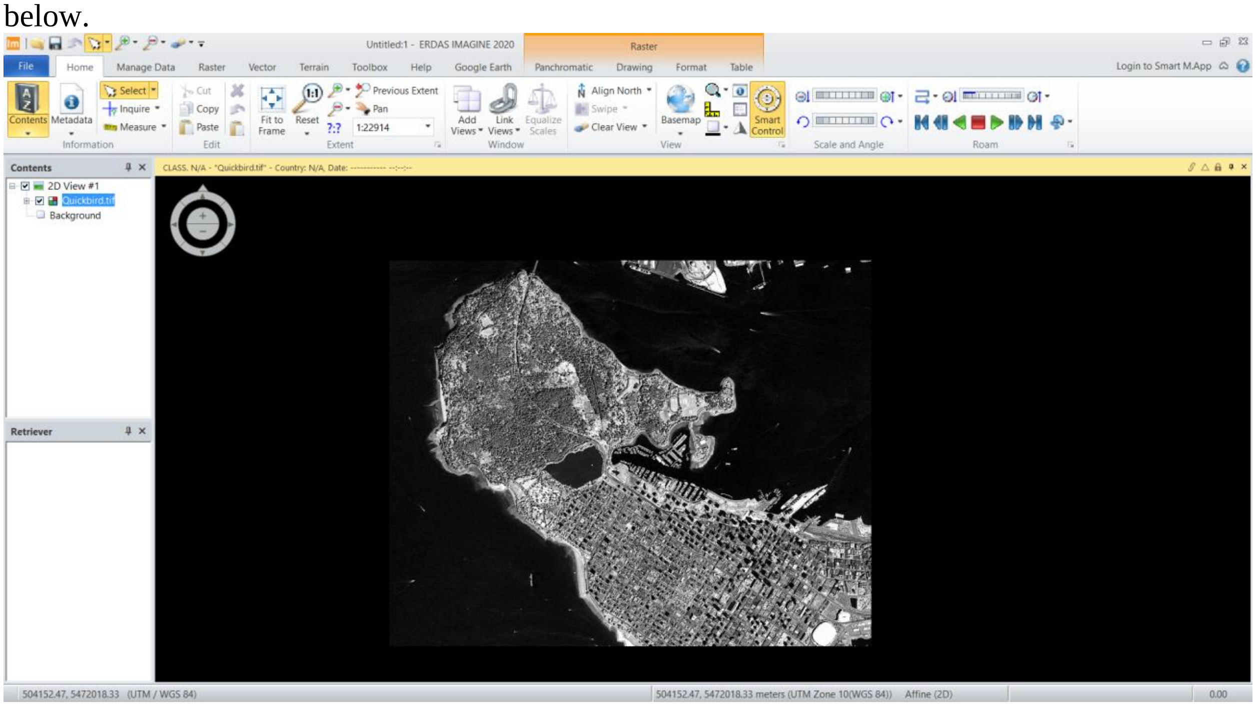
1. To load this image, in the main menu bar, go to File > Open > Open Raster Layer and then navigate to the place on your computer where you have saved the files associated with this lab.
2. Once selected, the image will appear in the main window, and the layer details will appear in the **Content Pane** area. Alternatively, you can right-click on the main **View** and select **Open Raster Layer** option and navigate to the place on your computer where you have the files associated with this lab.

NOTE: If you cannot see the files after you navigated to the data folder, click **Files of type** drop-down menu in the **Select Layer To Add dialog box**. Select the **All File-based Raster Formats** option from the drop-down menu. Now you will see the related image files in the data folder.



If you want to move the image, click 

on the above toolbar. The display image shows below.

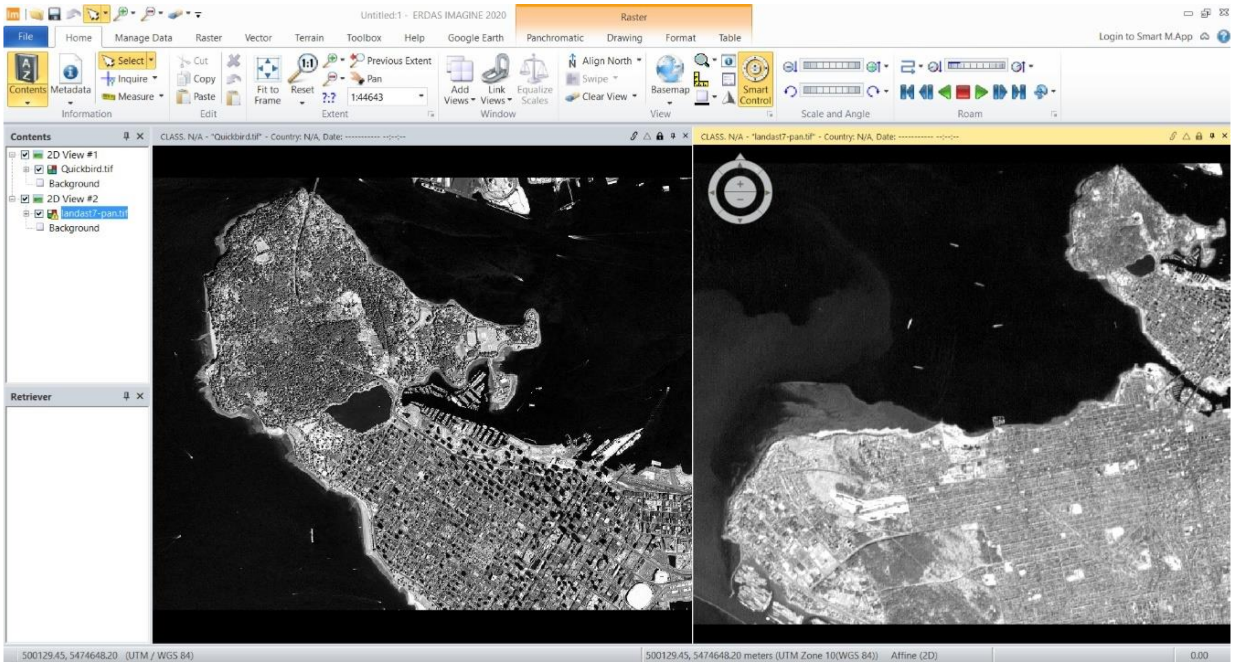


The second image to load is the "Landsat7-pan.tif".

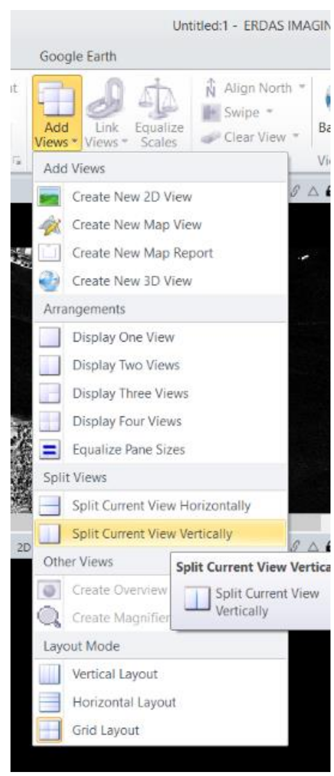
3. To do this, first go to **Home > Add View**. Then select Display Two Views. A second View will add to the Content pane.

4. Select the second View and then go to **File** and select **Open** > **Raster Layer**. Navigate to the data folder and select "Landsat7-pan.tif". Right-click both Views and select **Fit to Frame** option.

At this point, you should have both of these images (Quickbird panchromatic and Landsat7-pan) appearing in the Content pane, and both of these images displayed simultaneously on your screen.

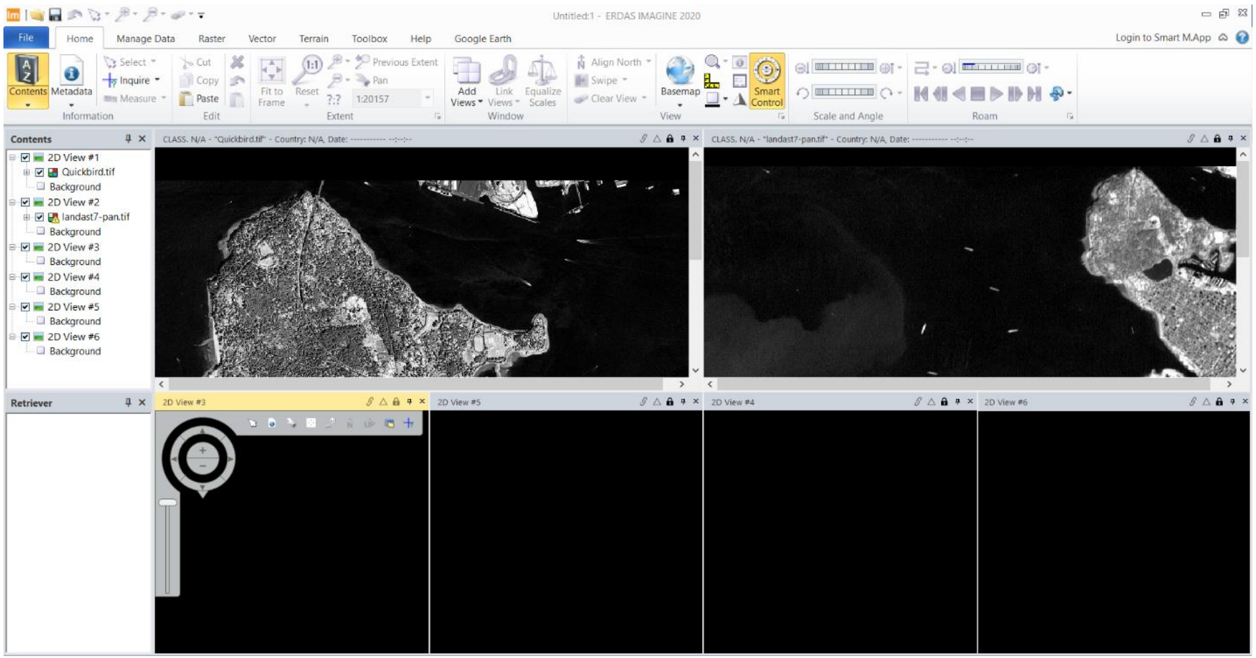


The second image is a panchromatic Landsat 7 ETM+ image of Vancouver. You shoul familiarize yourself with the characteristics of the Landsat ETM+ sensor by visiting NASA or Wikipedia webpage. Also, think back to Lab # 2, and remember the information provided regarding this sensor.



Now we are going to see four additional images; therefore, we need four more additional Views.

5. Go to **Home** > **Window** group. Select **Add Views** and a drop-down list will appear.
6. Select **Display Four Views**. Now you will see the main window with four windows where the bottom two windows are empty.
7. To add two more additional Views, we are going to split the bottom two windows into four Views.
8. Now, select **2D View#3** in the main window and select **Add Views**. Then select **Split Current View Vertically**.
9. Now you will see that a new window has been added.
10. Perform the same steps to **2D View#4** and add another view to the main window (sixth View: 2D View#6).



11. Go to the third window and right-click. Then select **Open Raster Layer** and navigate to the data folder and select **"Landsat99.tif".**

This time, you can see the image is colored, different from the first two. This is a "true-color" composite image taken at the same time as the panchromatic image that we just looked at. Remember what true-color means in relation to the last lab. To know more information about images, you can right-click the image name on the **Content pane** and **Select Metadata** to see the details of your photos.

12. The fourth image to open is the "Landsat85.tif" (this will result in View # 4), and you can follow the same steps as previously to open it.

This is also a true-color image from 1985, which means that the displayed colors should match the recorded colors by the sensor. This image has roughly the same spatial resolution as the previous image.

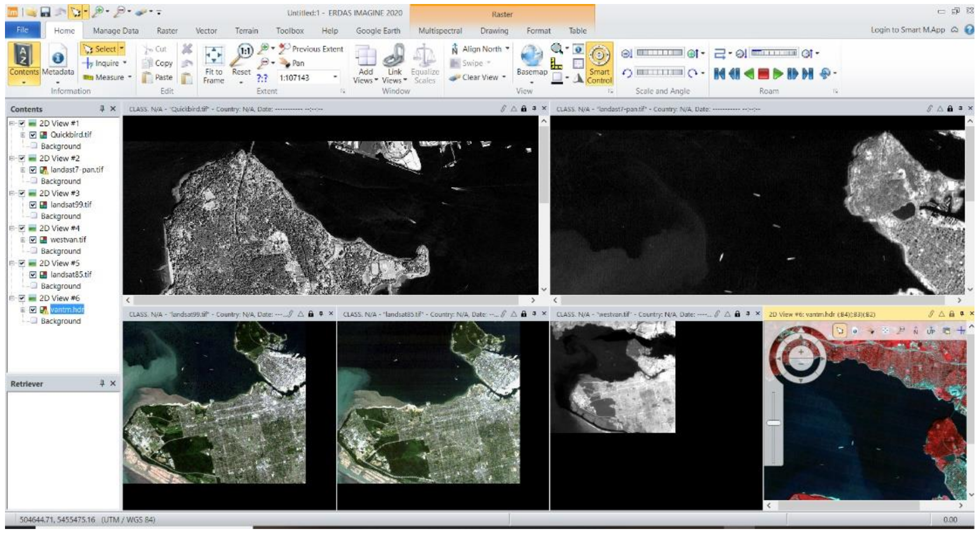
13. Next, you will load the "WestVan.tif" Image to View window (View#5), which has a spatial resolution of 60 m. This image has a coarse (comparatively) spatial resolution because of to the part of the spectrum that it is representing.

14. Up until this point, we have been looking at .TIF images. The next (and last) image we will look at is an ENVI formalt Landsat Thematic Mapper (TM) image of Vancouver from 1984 (vantm.hdr → View#6).

NOTE: This image is stored in ENVI's native format and consists of two files: a flat binary file without embedded non-image data, and an ASCII header file containing all of the information needed to open the file. The image consists of 512 lines and 512 cells (columns), forming a grid of 512 x 512 pixels (from picture elements). The image file has seven bands (channels), identified as bands 1 through band 7. The ground resolution of each pixel in bands 1 – 5 and 7 is approximately 30 x 30 m. For band 6, the ground resolution is roughly 120 x 120 m.

At this point, you have had open all the images for the first part of this lab exercise.

You should have a main window similar to this figure.

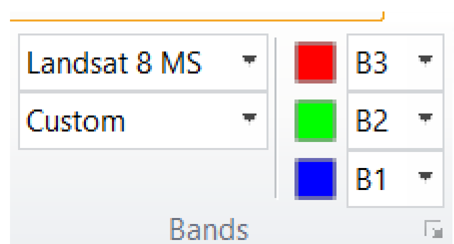


Q1 (10%): Include a snapshot of the main window with all the six images opened.

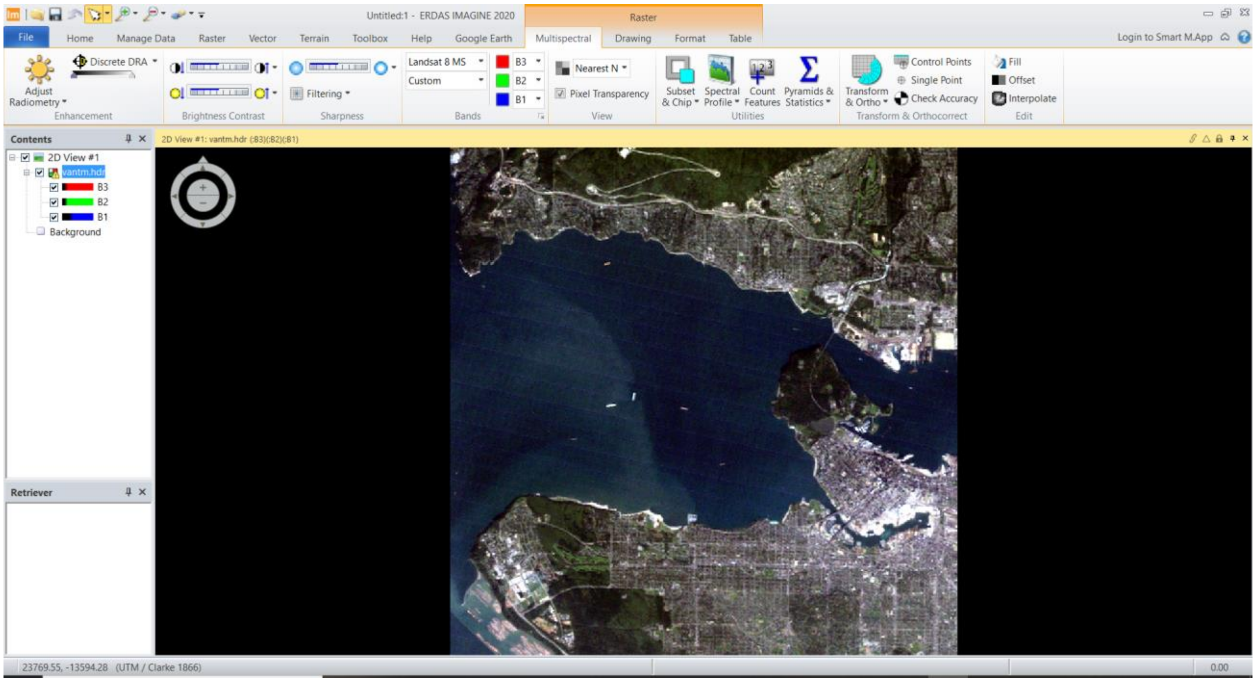
Next, close all the Views in the main window that you have had opened for the first part of the lab.

For the second part of the lab, please follow these steps:

3. Go to **File** and select **Recent** option. Now you will see all the recent image files you had opened.
4. Select "vantm.hdr" and open it.
5. As before, the file's bands will be available in the **Content pane**.
6. Once the image is added to the main View, the **Raster** Tab will activate. Select **Raster | Multispectral** and use **Band Selection** to select RGB Color. Assign the bands as follows:
   a. Band 3 to R
   b. Band 2 to G
   c. Band 1 to B



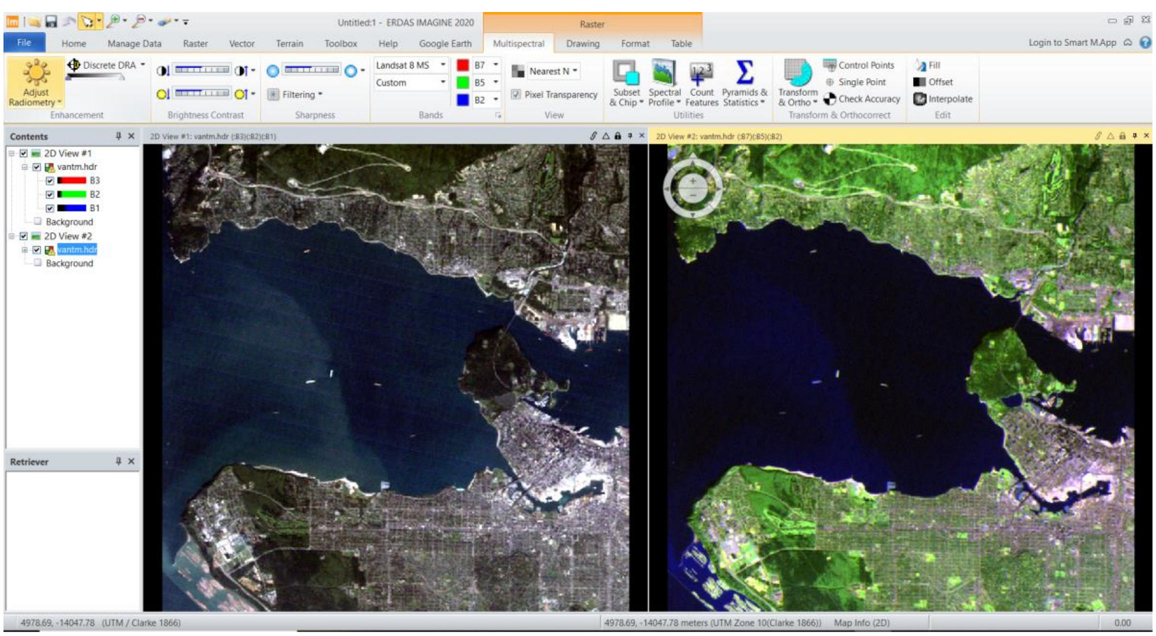
A color-coded three-band image of the Vancouver area opens similar to the following:



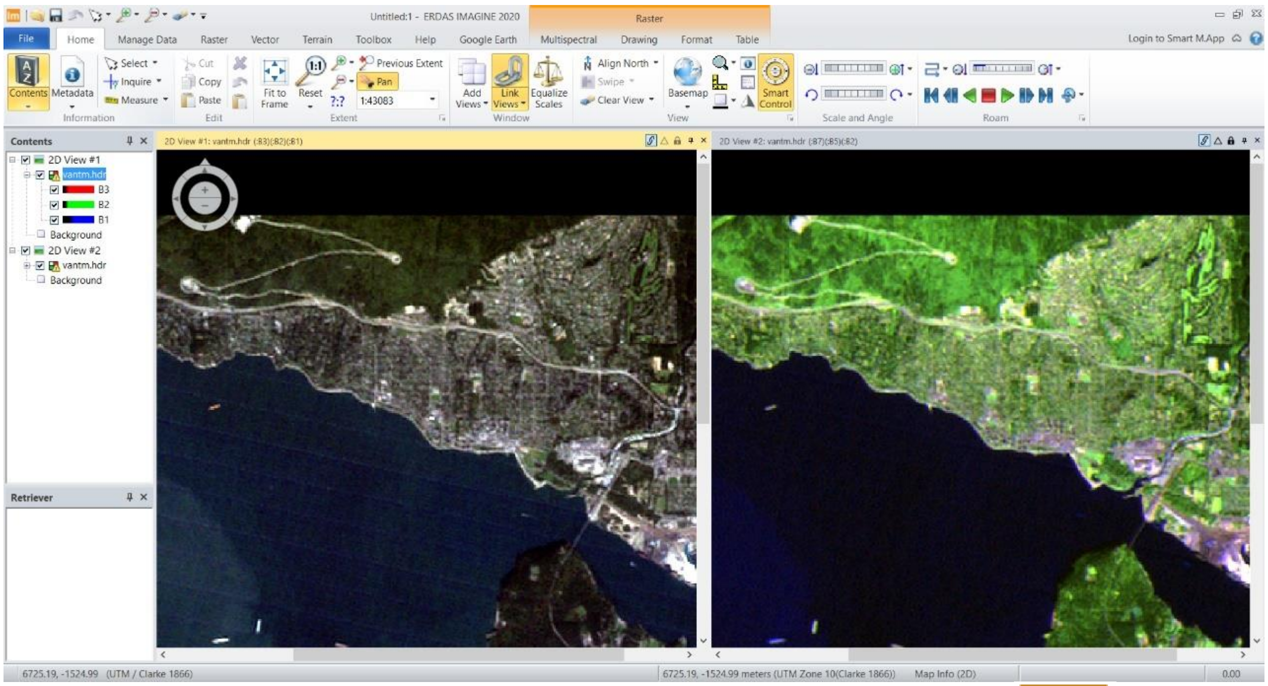
3.5.4 Making false color composites

Add “*Vantm.hdr”* to a new view. This time you will create a false color composite by selecting different wavelengths to be displayed as red, green, and blue.

1. Try to assign this band combination: **Band 7 → R, Band 5 → G, Band 2 → B**



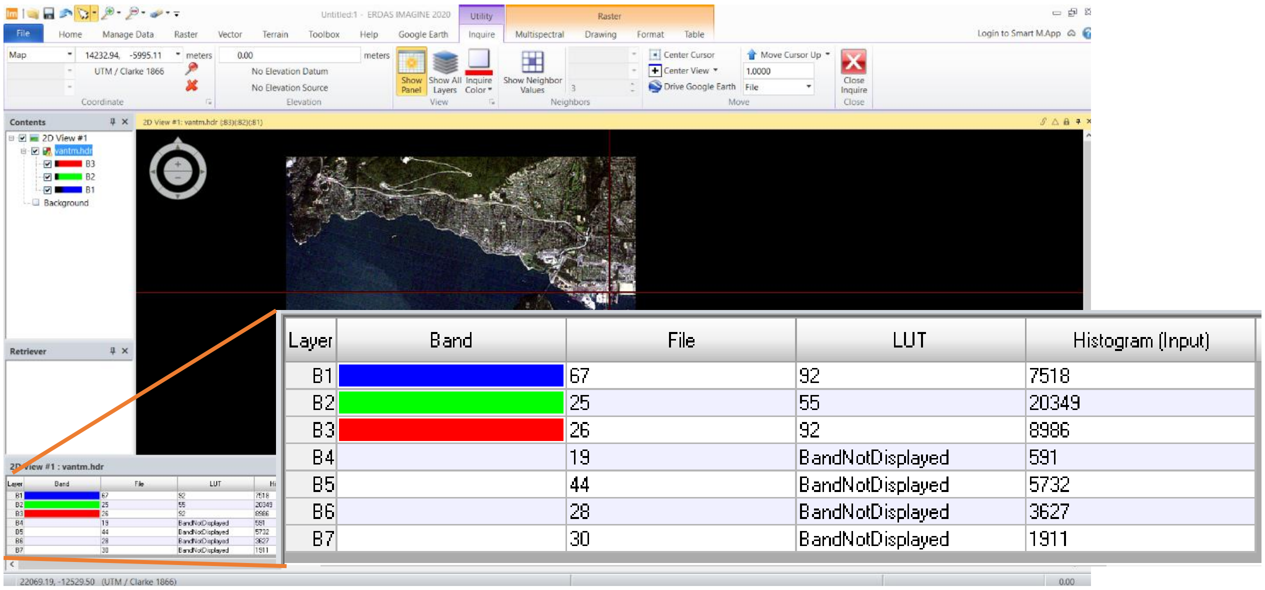
You should have two display open, One for the true color composite and one for the false-color composite. You can experiment with different band combinations by loading different bands on the Red, Green, and Blue boxes.
NOTE: A standard false-color composite, for instance, has Band 4 on Red, Band 3 on Green, and Band 2 on Blue.
You can also link your display groups so that navigation in one is matched in the others. From one of the displays, select **Link Views and Sync Views** options available under **Home** Tab. Also, Select **Equalize Scales** as well form window group under the **Home** tab.



Try navigating around the image in one of the display groups by using the pan  option. Once you click the **Equalize Scales** icon, the movements will be matched in the other display groups.

3.5.5 Displaying individual spectral bands

1. Remove **View#2** from the **Content pane** and assign true color configuration to the image in **View#1**.
2. From the **Home** tab, select the **Inquire** option. Now you will see **Inquire** tab is activated. This tab contains tools for measuring raster values at a specific point and helpful for finding the coordinates of a point on a raster.
3. Now, look at the attribute table at the bottom of the main window. This table shows the band combination of the current image display. More importantly, it shows the pixel values of the point (where the inquire point located) and their affiliated details.



NOTE: Elements of the Attribute table

Band: Displays which layers that are assigned to the red, green, and blue color channels

FILE PIXEL: Displays the actual pixel value from the image file.

LUT VALUE: Displays the lookup table value for the pixel.

HISTOGRAM: Displays the number of pixels (histogram) in the image that have this pixel value.

Lookup table: ***IMAGINE and Lookup Tables***

It is rare that the file values that are stored in a raster image can be used as brightness values.

- In image files of interval or ratio data such as remotely-sensed data, the file values are often low and too close together to create an image with discernible contrast.
- In image files of normal or ordinal data, the file values do not relate to brightness or color, but to a class number.

The transformation of file values into brightness values in IMAGINE takes place with the use of lookup tables. A **lookup table** is an ordered set of numbers, which is used to perform a function on a set of input values. To display or print an image, lookup tables translate file values into brightness values.

For example, the following lookup table translates numbers from the range 30 to 40 to the range 0 to 255:

30	>>	0
31	>>	25
32	>>	51
33	>>	76
34	>>	102
35	>>	127
36	>>	153
37	>>	178
38	>>	204
39	>>	229
40	>>	255

Three lookup tables are used in IMAGINE -- one for each color filter or gun. The source and function of the lookup tables depend upon the type of image displayed.

- In true color and grayscale images, lookup tables are viewed and edited using contrast adjustment.
- In pseudo color images, lookup tables are stored as a color scheme.

Q2 (10%): Include a screenshot of your main window that includes the attribute table.

4. Now navigate to Raster | Multispectral and change the band combinations as follows: **Red → B1, Green → B1, and Blue → B1**.
5. Observe how the image and attribute table changed. Because now you are displaying band 1 for the image.

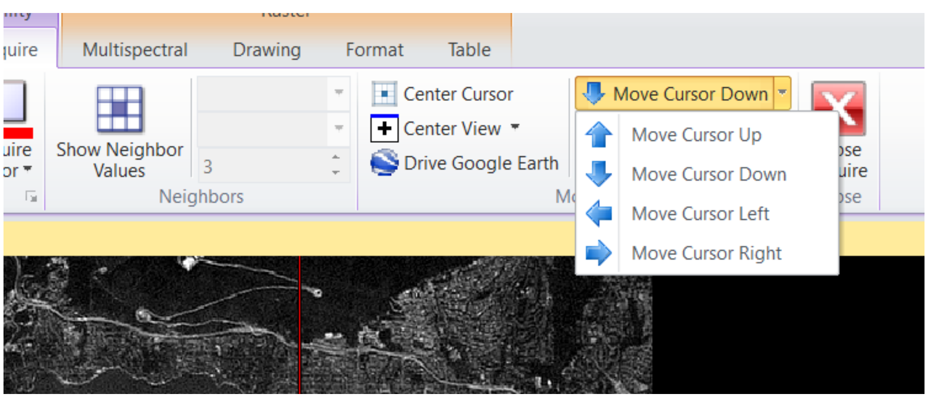
Q3 (10%). Include a screenshot of your main window that includes the attribute table.

6. You can move your inquire cursor by using the **Move options** available under the **Utility | Inquire** tab. Observe changes of the File, LUT, and histogram values by moving the inquire cursor.

3.5.6 Viewing image coordinates and digital numbers: DN

For many applications, you may need to know the image coordinates (pixel and scan in the case of raw images, UTM coordinates in the case of georeferenced images). These coordinates can be displayed using the **Inquire** tool. Similarly, the value of digital numbers at any given point (based on cursor position) is also displayed.

4. Select the **Inquire** tool.
5. Move your cursor over the window in the display and examine the changes in DN values and coordinates. You can use the **drop-down menu in the Move group under the Utility | Inquire tab.**

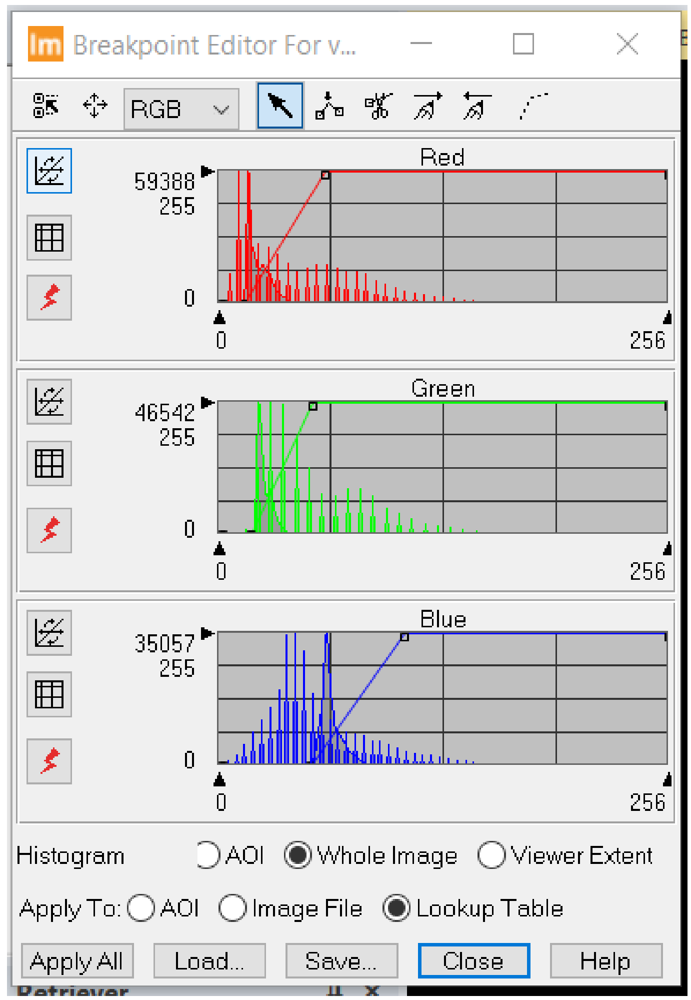


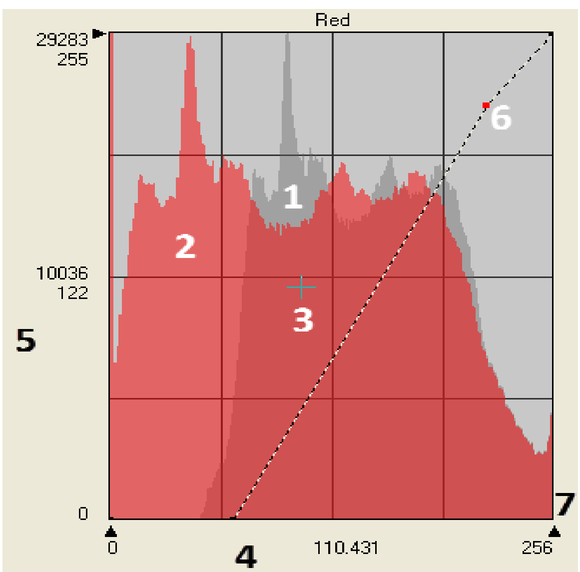
6. You can use the **Coordinate Group** in the **Utility | Coordinate** tab to observe the coordinates of the selected pixel.

3.5.7 Histograms

In ERDAS, we can use **Breakpoint Editor** to view, edit, and rescale the histograms and lookup table graphs for the red, green, and blue.
We can perform this when we have a color image displayed on the main View.

1. Adjust "vantm.hdr" image band combination to true-color configuration (refer step 4 on page 6). You can select Breakpoint Editor by right-clicking the true color layer on the **Content pane** > **select Breakpoints.**
2. Now **Breakpoint Editor For vantm.hdr** dialog box will open. To get a better look, you can maximize this dialog box.
3. Let's look at some features available in each graph.





1	Gray histogram - histogram of data values of the input image
2	Red histogram –histogram of brightness values of the Display device, in the color corresponding to this band. Can be Red, Blue, Green for multispectral images, and Yellow for panchromatic images.
3	Pointer - location of the cursor
4	X axis – Range of data values sampled in the statistics. 0 - Minimum data value; also darkest values of a continuous image 110.431 - Data value (X-value) at pointer location in histogram 256 - Maximum data value; also lightest values of a continuous image
5	Y axis – Contains two values because there are two histograms. Gray histogram - Range of frequencies of input data values is 0 - 29283 (upper number). Red histogram - Range of Display output brightness values is 0 - 255 (lower number). 0 - Minimum Display output brightness value 10036 - Data frequency value (Y-value) at pointer location in histogram 122 - Display output brightness value (Y-value) at pointer location 29283 - Maximum frequency value 255 - Maximum Display output brightness value
6	Breakpoint - Drag to define segments of the Lookup Table and to move these segments with the mouse.
7	Rescaling arrows

(Source: ERDAS IMAGINE help)

4. Click on **Add breakpoint** **icon** and you will see your cursor changed into + mark. Now you can click on the graph (lookup table) where you want to add a breakpoint.

NOTE: If you need to delete a breakpoint, you can use **Delete breakpoint** **icon.**

5. Drag your point, and you will see the changes in the lookup table graph. You can use **Apply LUT** icon in the histogram section or the **Apply All** button in the dialog box to see the changes you just made to the image.
6. Apply steps 5 and 6 for Green and Blue histograms as well and see how your “vantm.hdr” change.

Q4 (15%). Include a snapshot of your image along with the Breakpoint Editor dialog box.

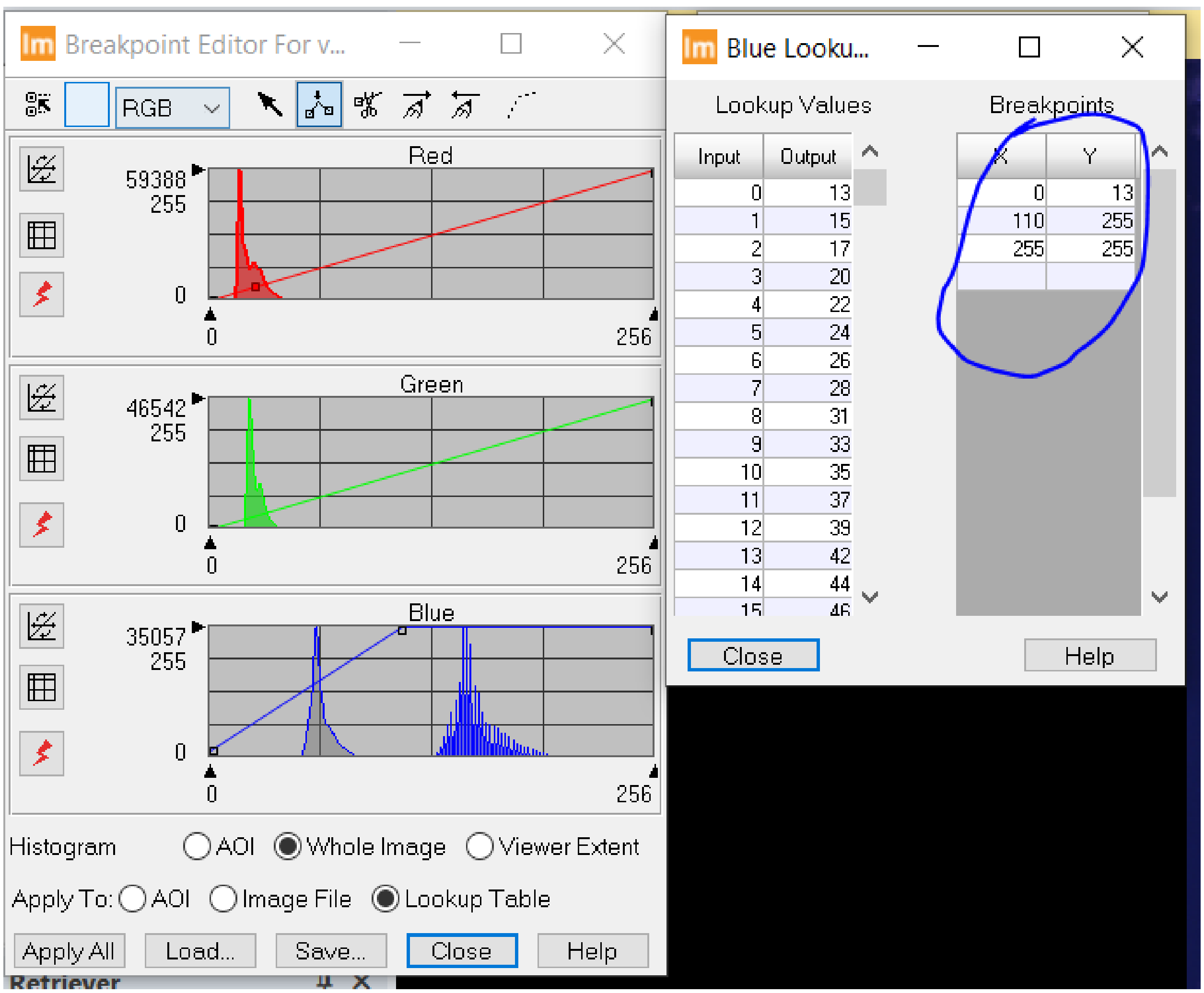
7. Now go to the red histogram, click **Lookup Table CellArray icon**. A separate dialog box will open, which is the lookup table editor for the respective color.

8. In this table, you can see two different sections, including **Lookup values and Breakpoints.**

NOTE: **Lookup Values table** has two columns, **Input** column that contains the pixel data value of the input data. The **output** column shows the brightness value of the display device. You can change the X and Y coordinates of existing breakpoints located on the lookup table graph in the Breakpoint Editor in the **Breakpoint table**.

9. You can enter different y values and see how those values affect the respective graph and overall image presentation. Press enter once you enter a new value and click **Apply LUT** to update the changes.

Q5 (15%). Include snapshots of the Lookup Table for each band along with their Lookup table graphs. Also, include a snapshot of your main View after you apply all the changes.



Input	Output
0	13
1	15
2	17
3	20
4	22
5	24
6	26
7	28
8	31
9	33
10	35
11	37
12	39
13	42
14	44
15	46

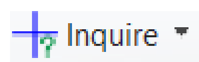
X	Y
0	13
110	255
255	255

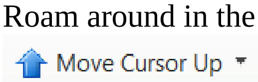
3.5.8 Image enhancements

The pixels from an image can have any value: negative, positive, integer, or floating-point. When the image data are visualized on screen, they are displayed as brightness values for each screen pixel. A data pixel with a larger value will be brighter than one with a smaller value. However, unlike the image data, screen pixels can only have 256 unique brightness values, varying as integers between 0 and 255 (where 0 is black and 255 is white). Clearly, this limitation prevents the data from being displayed with brightness exactly equal to their real value. For example, how would you display a negative data pixel or floating-point data ranging between zero and one?

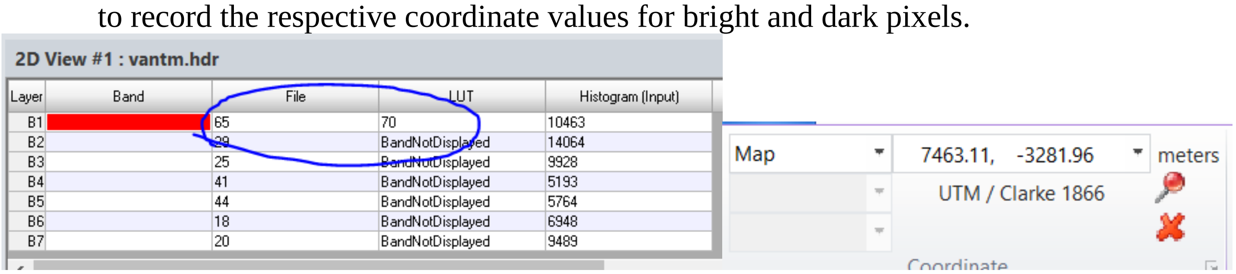
Stretching the image data refers to the method by which the data stored by each pixel is rescaled from original values into a range that the monitor can display — namely, into integer values between 0 and 255 (if the monitor has a color depth of 8 bits). For example, if the image data were floating-point values that ranged between -1.0 and 1.0, the image might be stretched such that data values of -1.0 are assigned a brightness of 0, and data with values of 1.0 are assigned a brightness of 255. All of the intermediate data values would be assigned new stretched values based on a simple model or *stretch type*.

Commonly, a linear stretch type is used so that the stretched data values maintain the same relationship to each other as the original data (e.g., the relative distance between two stretched values is the same as the relative distance between the two original data values). Other stretch types use different models to assign intermediate values, such as gaussian, equalization, or square root functions.

7. Display the first band of the "*vantm.hdr"* image.
8. Open the **Inquire** attribute table by clicking **Inquire icon** 

.
9. Roam around in the display window using **Move Cursor up/down/left/right** option 

available in **Utility | Inquire** tab. Try to find some of the brightest and darkest pixels in the image (you can zoom in or out to use the resolution you think it is appropriate for you to operate).
10. Using the **Attribute table**, note the data value, the corresponding screen value (i.e., File Value and LUT value), and the image coordinates for one of the bright and dark pixels. **To answer the next questions, include a snapshot of the area used, attribute table and coordinate values:**
11. You can utilize options available in the **Coordinate** group under the **Utility | Inquire** tab to record the respective coordinate values for bright and dark pixels.

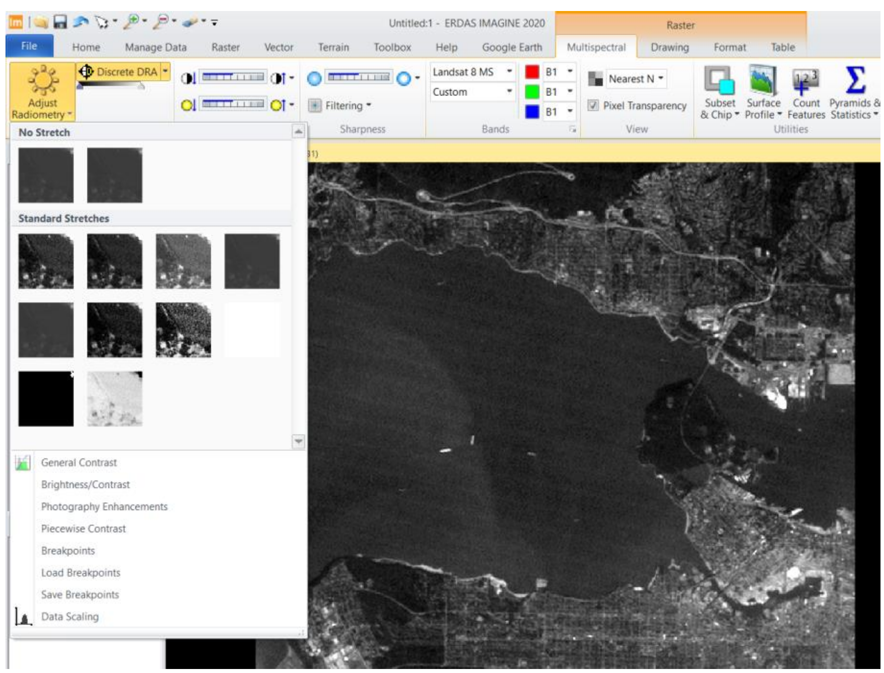


The exact method that is used to rescale the image data into brightness values can make a drastic difference in the way that the image looks. Thus, it is quite common to adjust the parameters of the stretch in order to maximize the information content of the display for the features in which you are most interested. This process is referred to as *contrast stretching* because it changes contrast in the image -- the relative differences in the brightness of the data values (i.e., increasing contrast means that the dark pixels are darker, and the bright pixels are brighter, so the brightness difference between the two is increased.

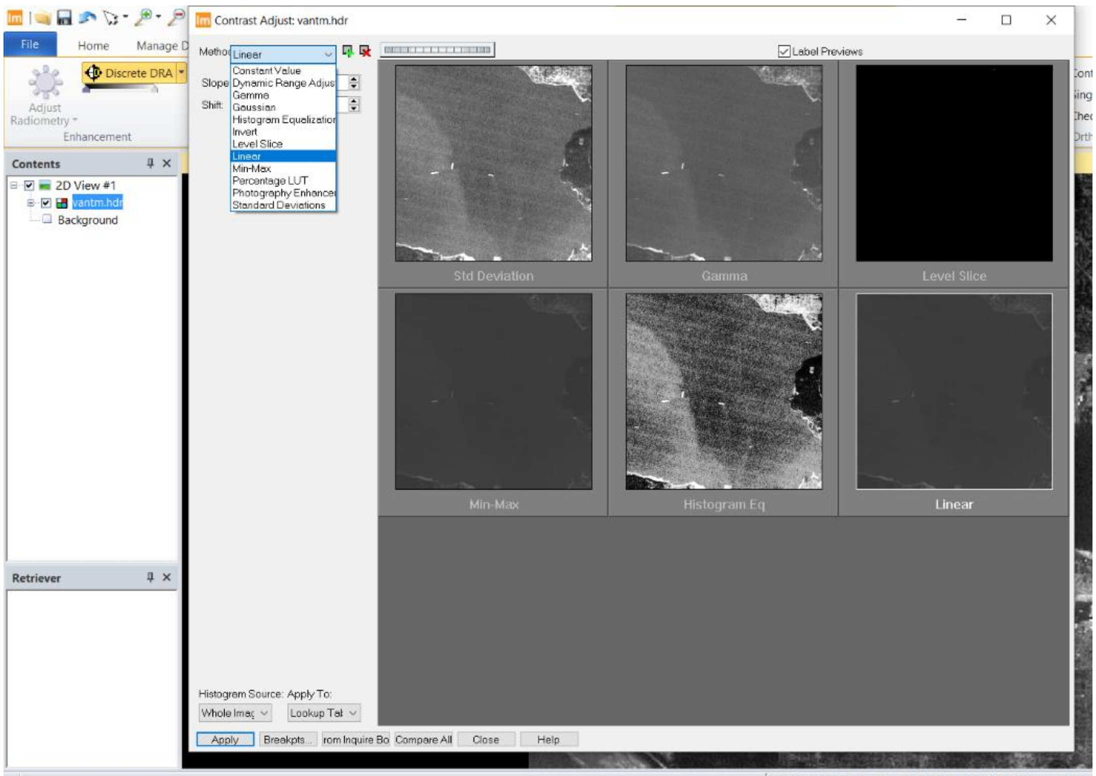
For example, consider an image whose data numbers (DN) are integers that range between 35 and 85 (only 51 different data values). If this image was stretched with a simple “one-to-one” model where a data value of 0 is assigned 0 brightness, and a data value of 255 is assigned 255 brightness, then the image display will be quite dim (since the brightest pixel is only brightness of 85). This stretch produces a low contrast image because a difference in data value of 1 unit is represented by a difference in brightness of 1 unit. Furthermore, much of the range of available screen brightness is not being used because there are only 51 different values in the image data (i.e., there are no pixels with a brightness between 51 and 255, so this brightness are unused). The image contrast could be maximized by assigning a brightness of 0 to the minimum data value of 35, a brightness of 255 to the maximum data value of 85, and linearly stretching the remaining 49 data values through the rest of the available brightness range. This increases the contrast because adjacent data values now differ by several units of brightness rather than just 1, making it easier to visually distinguish slight differences in the data values.

Through careful adjustment of the image stretch, it is possible to highlight certain features in an image, and ERDAS provides several sophisticated tools for this purpose. ERDAS’s *default stretch* (defined in the configuration file) is Standard Deviation Stretch. By default, this process uses the data between -2 and +2 standard deviations from the mean of the file values and stretches those values to the complete range of output screen values.

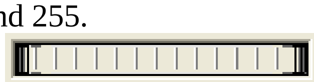
12. You can choose different type of stretching options by selecting the **Adjust Radiometry icon in Enhancement group under Raster | Multispectral** tab. Be familiarized with different starching options available under this drop-down menu.



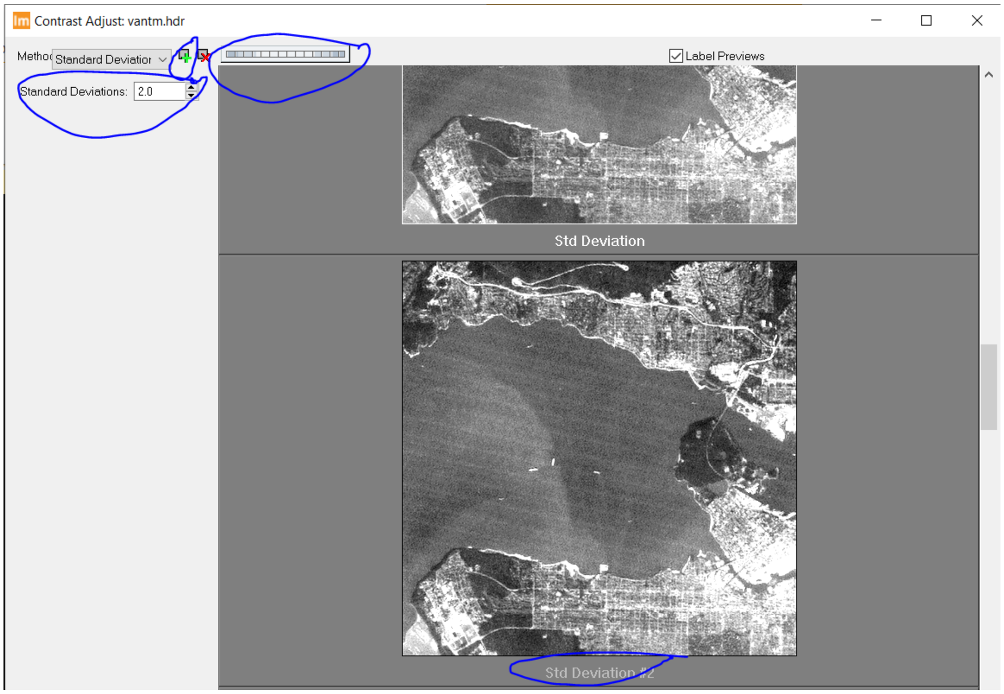
13. Also, you can select the **General Contrast** option from the **Adjust Radiometry** drop-down menu. A separate window will open and maximize the window size.
    Here you can select different starching methods, and all of them will add the **Previews window**. So, you can select the best option for your image enhancement.



Several of the commonly used contrast stretches can be applied to the displayed image without having to manually define the parameters for the stretch. The *Linear* stretch sets the data minimum and maximum to screen values (brightnesses) of 0 and 255 and stretches all other data values linearly between 0 and 255.

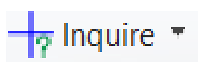
14. Now  Click and drag the zoom wheel to enlarge or reduce size of chips in **Preview pane**.
15. Select **Standard Deviation** from the drop-down menu. Then set **Standard deviation to 2**.
16. Click add new **Preview image**  icon, this will add another **Standard deviation preview (Std Deviation #2) to the Previews window.**
17. Change the **Standard deviations** to a lower number (i.e., 1) and compare it with previous image (Standard deviation =2).

Q6 (10%). What features does this stretch highlight (by lowering the standard deviation value)?



18. Select the image with Standard Deviation 2 and click apply. This will add to the main View and Minimize the **Contrast Adjustment** dialog box.

Find an area in the image with the very brightest pixels that is nearly saturated with white pixels). Zoom in on this white patch so that the zoom window contains mostly saturated white pixels.

19. Using the **Inquire icon** 

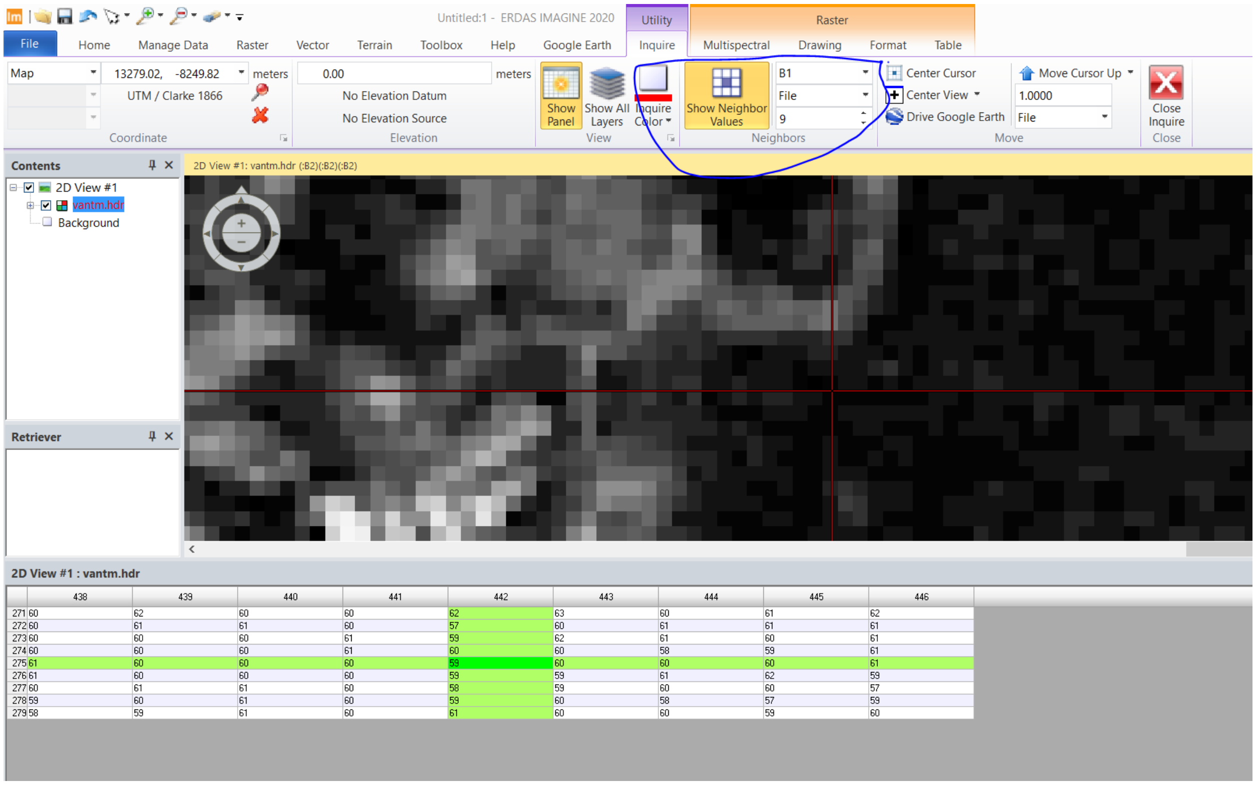
, note that even though the pixels in the Zoom window all appear to have the same brightness, they actually do have slightly different data values (refer to the attribute table)

20. You can use **Show Neighbor Values** option to study the brightness values in adjacent pixels. To perform this task, you can select **Show Neighbor Values** icon in the **Neighbors group under Utility | Inquire.**

21. Select Center cursor and setup Neighbor **Row = B1, Neighbor column = File, and Focus value to 8.**

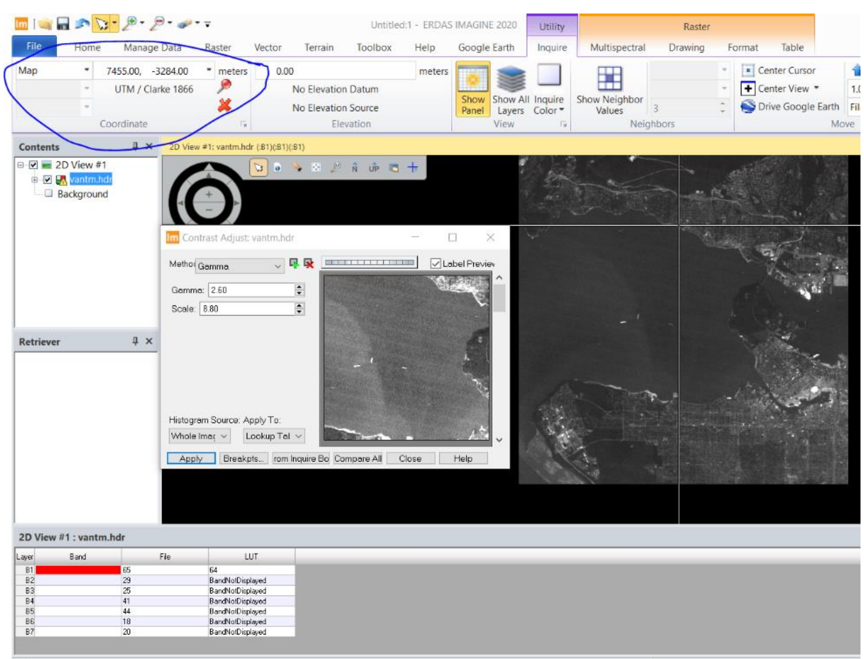
22. You can see the attribute table results at the bottom of the main View. Your selected pixel will be highlighted.

Q7 (15%). Include a screenshot of the main View that includes the attribute table.

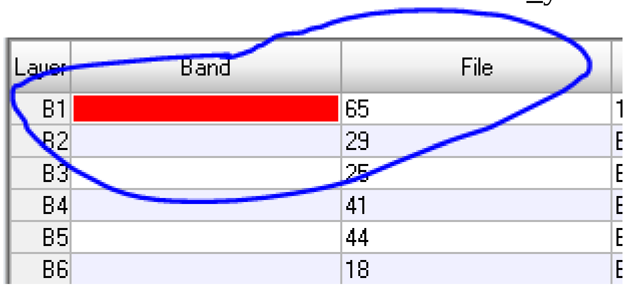


Contrast stretching an image for display does not affect the original image data. It is important to remember that whenever numerical processing is performed; the original data from the file are used, not the contrast stretched data that are displayed.

1. Now **Restore down the Contrast Adjust dialog box** and setup you screen similar to following setup. Also deactivate the **Show neighbor Values** tool as well.



2. Navigate to the **Coordinate group** and enter the coordinate values you have recorded from step 4 in section 3.6. You will see that **Inquire cross** hair will move to the same exact place you had previously in section 3.6.
3. Now go to Contest Adjust window and change constant adjust methods one by one form the Methods drop down menu. Each time click apply, so you can see the change of image enhancement from the main window.
4. Save your work. File > Save As > Session > Lab3_yourname.



Q8 (15%). Compare the data (in File column) you recorded in Section 3.6 with the data recoded from above step (step 3). Do you observe any changes when you changing different stretching methods? Explain your answer. Use appropriate screenshots if it is necessary.

Please submit the answers to questions on Canvas as a pdf file. Also include the question and question number with your answers.

4 Stereopsis & Photo Typing

Lab developed by Dr. Strimbu from the notes of Dr. Hilker.

4.1 Objectives

- Use stereo vision
- Interpret orthorectified images
- Estimate areas on an orthorectified image

4.2 Stereopsis

4.2.1 Overview

When aerial photographs are obtained in sequence by an aircraft traveling at a defined height and speed, the characteristics of an object can be measured in three dimensions. The capacity to measure objects when seen from different positions (perspectives) is using the perception of depth. The ability to identify 3-dimensional structures from binocular vision is called ***stereopsis***.

Stereoscopic coverage is obtained where two photos overlap. If your left eye looks directly at an object on the left-hand photo, while your right eye looks directly at the same object in the right photo, your brain puts together a 3-D image of the object because of the difference in the two angles of view.

A **stereoscope** is used to achieve the required optical isolation of each eye, as well as to provide magnification for air photo interpretation. Because the distance traveled by plane between the times the two photographs were taken greater than the distance between your eyes, the vertical dimension is greatly exaggerated in aerial photography, and slopes can look like precipices.
A pair of successive overlapping photographs along a flight line constitute a *stereo pair*. A stereo pair may be viewed with a stereoscope to produce a three-dimensional representation called a *stereo model*.

How to see Stereo?

1. Select a pair of photos and make sure they form a stereo pair (provided by the instructor).
2. Look for a feature which you can see in both photos. Place the two photos side by side on the table under a stereoscope, make sure that they are lined up correctly along the line of flight. They should be spaced apart approximately the same distance as the lenses of the stereoscope.
3. Move the photos gently around beneath the stereoscope until the images come into focus, and the illusion of three dimensions is created.

The following method may be useful to achieve a stereoscopic view.

1. Put the tips of your index fingers on the same object in the left and right photos.
2. Move the fingers until the fingers are fused, then remove fingers and leave photos intact.

Tips:

1. Make certain that the photos are properly aligned, preferably with the shadows falling toward the viewer.

2. Keep the eye base and the long axis of the stereoscope parallel to the flight line.
3. Maintain an even glare-free illumination on the prints or transparencies.
4. Arrange for comfortable sitting and sufficient illumination.
5. Keep the lenses of the stereoscope clean, properly focused, and separated into your inter-pupillary distance.

Select a set of photographs and attempt to see them in stereo. Take your time and relax. If you cannot see stereo in the first attempt, select a new set of photos and try again. If you still have trouble select an urban stereo pair, or one with topographic relief, as they as sometimes easier to see on stereo.

4.3 Visual Image interpretation

4.3.1 Tree and Tree Species Identification

Look at the sets of prints of individual trees. The photos have been taken on a very large scale (1: 600). The prints cover several common tree species in Oregon (Figure 6).

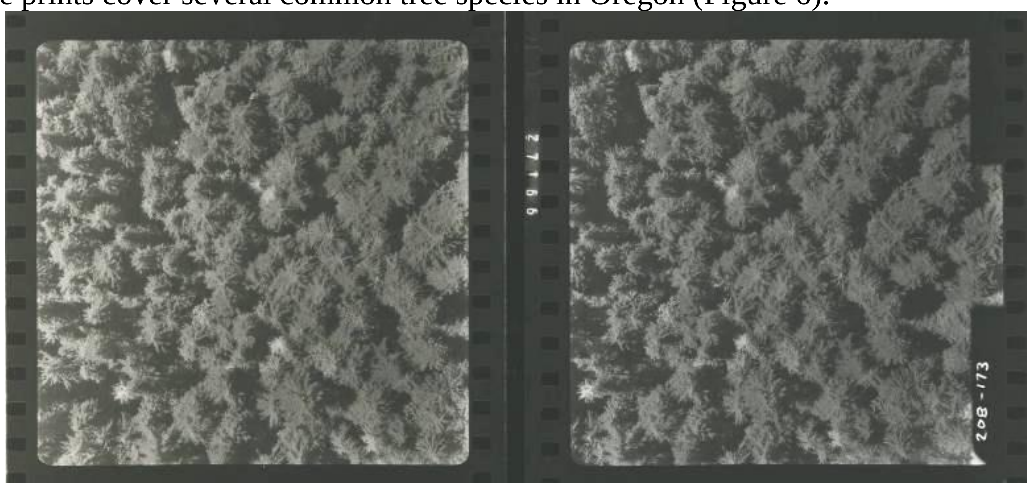


Figure 6. Stereo pair from Oregon

Q1.[10%] Different features: List four (4) different features of the trees in the scene.

No.	**Feature**
Example	Size of Crown varies between species
1	
2	
3	
4	

Species identification.

Try to identify different tree species visually based on the features shown below. How do you think the tree crown of a conifer might look like as opposed to an Alder tree? Where would you look for Alder trees? Try to identify a few unique features about the tree species

Q2. [20%]In the table below, complete two more features for each crown.

Species	Feature 1	Feature 2	Feature 3	Feature 4
Alder	Net shaped pattern	Smooth texture		
Douglas fir	Arrow shaped branches	Tall		
Western Hemlock	Skinny branches	Lots of branchlets		
Cedar	Spindly crowns	Lighter in tone		

4.3.2 Stand delineation

A forest is comprised of individual stands. A stand is formally defined as a managed ecosystem with trees as the main species. In most instances, age or species composition will differentiate stands. An area with hardwoods would be constitute a different stand than an area with conifers. Similarly, an area with 100-year-old trees would form a different stand than an area with newly planted trees. Theoretically, stand delineation is objective, but in practice, different people may look at the same forest and draw different stand boundaries, which make it in fact a subjective process. For instance, consider a mixed conifer stand forest with a small pocket of hardwoods in the middle. Is the pocket of hardwoods a separate stand, or is it incorporated as part of a larger, mixed-species stand? In many instances, these questions often come down to personal preference. However, operationally a stand is defined strictly by the organization managing the land, such as a stand cannot be less than 1 ac, for example. This choice is defined by the smallest areas that can be managed independently. In the case of the pocket of hardwoods in an otherwise coniferous stand, is the pocket large enough to support its own harvesting operation? If yes, you might consider this a separate stand. If not, it might be better incorporated as part of the larger, surrounding stand. For accurate stand delineation, it is advisable to look at aerial images as well as walking thru the forest and examine it on the ground.

Delineation of stands, or boundaries identification, is a subjective process, but it is advisable to start at a known location within your forest, such as an intersection of roads, a house, or a pond. Next, look for areas that have the same tree species composition, basically, shape and color, similar ages, which means the same texture. Using the main visual interpretation tools mentioned in the lecture and in the textbook, identify areas of the forest look the same and what areas look different.

Q3 [10%] What are the main tools used in image interpretation?

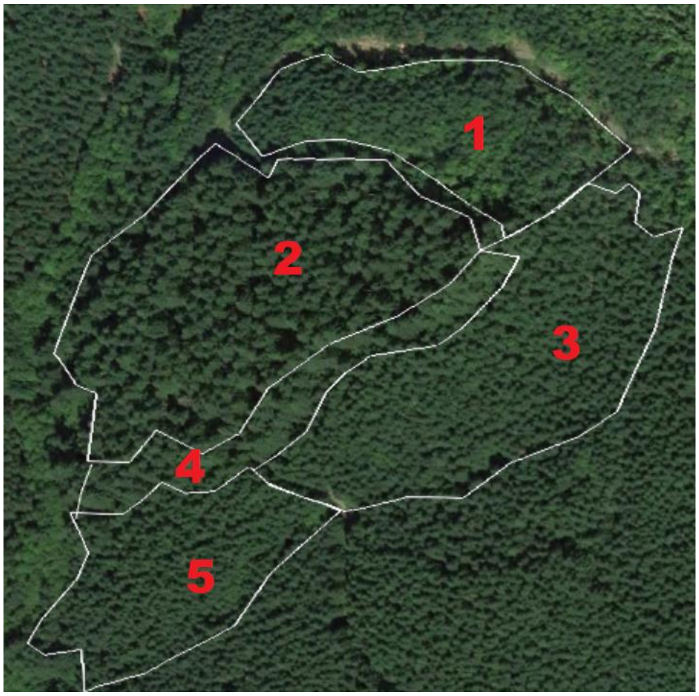


Figure 7. Stand delineation following visual image interpretation

Figure 7 shows the stand boundaries drawn on the aerial photo, which can now serve as a stand map. Notice the difference in color and texture between the different stands.

Once you have identified your stands, give each one a name to identify it. There are two common systems of naming stands: one that is database-driven, on which each stand is named with a string of letters and numbers meaning something from a data management perspective (such as **A1A1, A1A2**, etc.), and one that is personally driven, such as the stand name means something to the user or owner, such as **John_DF2011LakeNorth**, meaning the stand of Douglas fir planted in 2011 north of the lake on John Road.

1. Choose a forested area of approximately 1 square mile around Blodget, OR from Google Earth.
2. Delineate at least 5 adjacent stands using the **Add polygon** tool  available in Google Earth.

Q4 [5%] Include a snapshot of the map with the boundaries of the stands.

Note: You could enhance your polygons by choosing a 100% transparency and a boundary line thicker than the default 1 point. You can get access to these features by right-clicking the polygon and selecting the properties option. Then you will see the Google Earth- Edit polygon dialog box where you can change transparency and line thickness.

4.3.3 Estimate area of a stand

In the field, it is helpful to estimate the acreage of a stand from an orthophoto. Usually, this is not exact, but a reasonable estimate should work fine in the field for the purpose of course estimation of resources. The current technology allows you to measure areas with various GIS software on portable devices, such as Google Earth, and lately, Adobe has a tool that lets you measure the acreage inside the pdf file. Nevertheless, in many instances, such as lack of battery or power supply, you could be tasked to estimate the area of a stand in the absence of software, using only the aerial orthophoto. In these situations, the easier way is to use a dot grid.

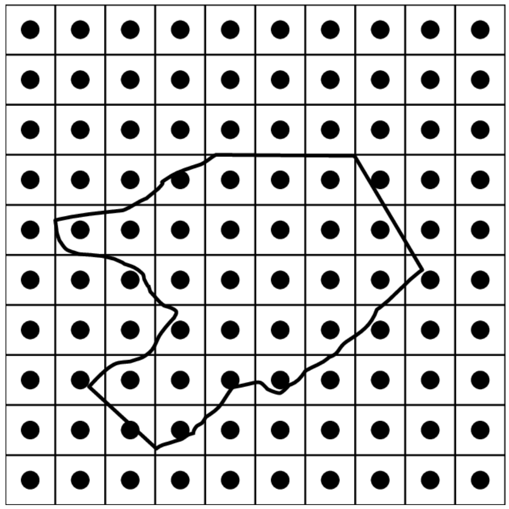


Figure 8. Area of a polygon estimated using a dot-grid (image from Koester et al. 2004. Sweep Width Estimation for Ground Search and Rescue)

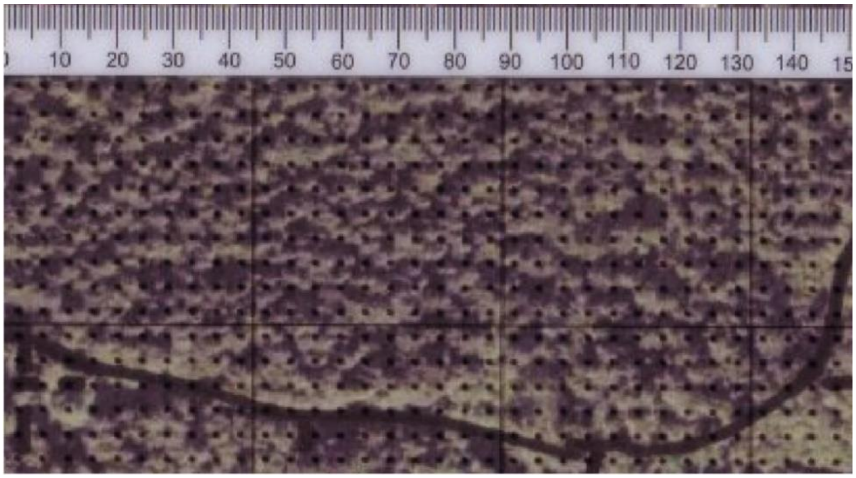
A dot grid is a series of evenly spaced dots (Figure 8), often printed on a clear plastic sheet. These grids are available from forestry supply merchants. You can also make your own using a piece of graph/grid paper. The method of measuring the area is very simple and relies on the scale of the orthophoto. The procedure is very simple:

1. Overlay the grid on your map/photo,
2. Count the squares or dots that cover each stand,
3. The area is simply the product between the number of dots within the stand with the area represented by each dot.

The first step is either to acquire a grid or to create one. If you have a dot-grid, then you can skip the creation of a grid part, but if you don't, you should continue reading. A grid can be created easily using a transparent sheet of plastic (provided) and an engineering paper. Overlay them, with the clear sheet on top, then with a sharp pen, pencil, or needle mark the corners of the engineering paper.

The dots would cover a square with at least 100 dots, meaning 10 x 10. The next step is to determine the area represented by each dot. To do so, you should measure the distance between the dots then convert it to units. For example, if the distance between two dots is 1/8 inch and the scale of the map is 1:24,000, then the distance between dots is 3000 inches or 250 feet. Each dot represents a square with the side distances between the dots, as shown in Figure 8. Therefore, the area represented by each dot is the square of the distance between dots, in the example 6.250 ft^2 or 0.1434 ac.

Once you know how much area each dot represents, then you should count how many dots are



inside the stand.

Figure 9 shows a dot-grid overlaid on an aerial photo of a stand. Let assume that inside the stand, you have counted 214 dots.
The final step is to estimate the area of the stand, which is the number of dots multiplied with the area represented by each dot. Therefore, the stand area is 214 x 0.1434 acres or 30.4 acres.

Q5 [25%]. Print the Google Earth image with the delineated stands from Figure 7 (be sure that scale bar is included in the print). Using the dot-grid and the scale bar, estimate the areas of the five stands.

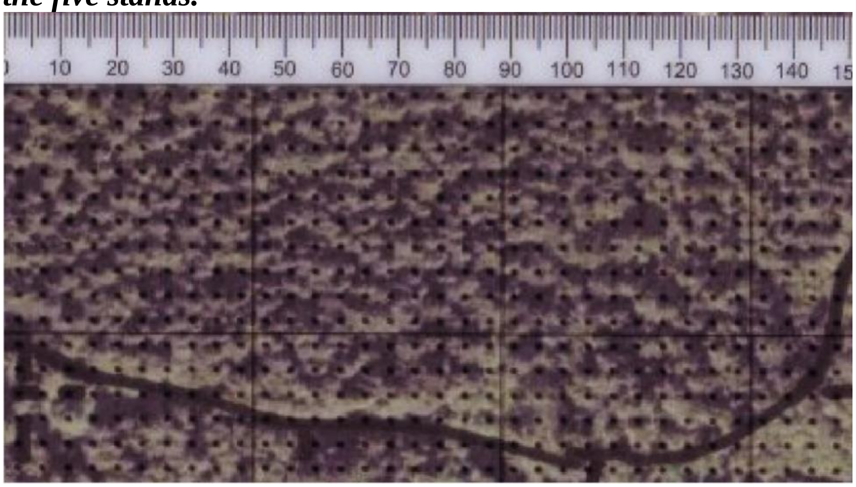


Figure 9. Dot-grid overlaid on an image on which the boundaries of a stand are drawn.

4.4 Landforms and Land uses in Google Earth.

In your lab, there is a .kmz file which contains questions that are tied to locations in Google Earth. Double click on the GoogleEarth.kmz file to start Google Earth. Then, double click on each of the questions, and it will bring you to the location that this question relates to. The questions are also printed below.

Q6 [5%] What do you think is happening in the forested areas at this location?

Q7 [5%] What caused this feature? (Zoom in with the little yellow person for a better appreciation of it)

Q7 [5%] What kind of housing is this (directly under and immediately surrounding the pin)?
Q8[5%] Starting at the pin and extending north along the coast for almost 6 miles, what do you think is happening in this area?
Q9 [5%] What large structures do the large grey roads provide access to? Approximately how large do you think they are (use things visible in photo to get a scale approximation). What do the smaller, tire track roads seem to provide access to? This answer is not as apparent as the previous one and requires consideration of where the image is located (the state should help).
Q10 [5%] What are the new features added to the river (clearly seen in 3/10/2005) that were not present in the 1994 photo? What do you think they are used for?

4.5 Advanced topics: Photogrammetric Point Clouds

Lab developed by Dr. Strimbu using data from Vernon Parish, Louisiana.

Objectives

- Basic operation with point clouds
- Generate Photogrammetric point clouds from airborne stereo images
- Knowledge on high-resolution aerial images

1 Files Used in this Tutorial

The lab will use four files, three aerial stereo images, and one a block file. The aerial images were acquired with an UltraCam X Camera, produced by Vexcel Imaging GmbH from Austria.

- The image files are **LA001_0003.tif, LA001_0004.tif, and LA001_0005.tif.** To ensure that the images are positioned correctly, another file is created during the photogrammetric process, which is not presented in this lab: the aux file. The aux file, which is an abbreviation from the auxiliary file, is used in conjunction with non-native ERDAS IMAGINE files accessed thru the dynamic library Raster DLL and stores information not normally supported by the .tiff file.

- The block file is: FE544LA.blk and contains the photogrammetric information of the images, particularly the Interior and Exterior Orientation Parameters, the Coordinate System, the Average Flying Height, the Sensor Model, and Camera calibration information.

Q1. [10%] What are the wavelengths recorded by the UltraCam X?
Q2. [10%] What is the pixel size of the UltraCam X sensor?

BEFORE YOU DO ANYTHING ELSE: *Retrieve the data for this lab from your course drive. The data for this lab can be found on T drive under Teach/Classes/FE444/2020/Lab4/. Download all the files locally, on the computer, as they are very large (more than 1 GB/file).*

2 Background

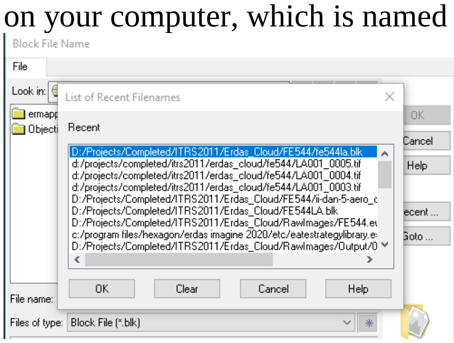
The remote sensing files stores spatial information in two formats: raster, which is based on matrices, and vector, which is based on points and relationships between points. Both types can store planar coordinates (i.e., x and y) as well as three-dimensional coordinates. When the 3D information is stored in a raster format, the files are commonly named digital elevation models, whereas, for vector format, the files are commonly referred to as point clouds. There are two types of point clouds, one obtained from passive sensors, like images, referred to as photogrammetric point clouds, and the other obtained from active sensors, like laser, referred to as lidar point clouds. Photogrammetric point clouds are obtained using stereometrics principles, which rely on the observation that the perception of depth is obtained by observing the same object from different viewpoints (locations). To make exact measurements using stereopsis, there are several details that must be known for each image, among which the following are the most important one:

- Image id
- Interior Orientation Parameters
- Exterior Orientation Parameters
- Coordinate System
- Average Flying Height
- Sensor Models
- Camera calibration information.

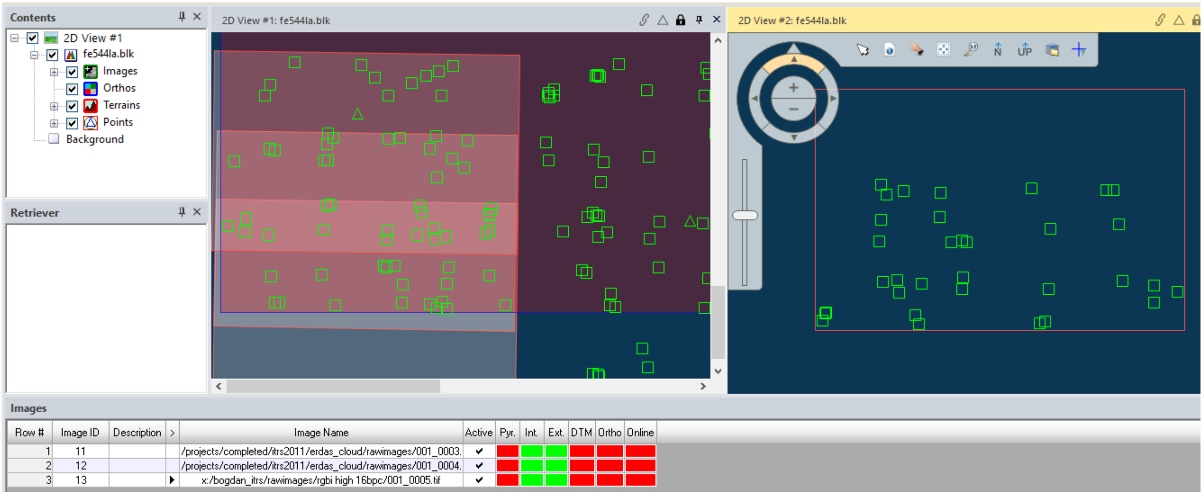
Once the images are properly aligned and positioned, dimensional measurements are possible, including elevation. The process of correctly position the images is mandatory and requires several steps, some iterative. Because of the length of the lab, the positioning of the images is not included. The file containing all the information needed for 3D measurements is called the block file (it has the extension blk), which is provided. If dimensions are needed in a raster format, then digital elevation models are produced from the stereo pair. However, if the vector type format is required, then point clouds are produced. In this lab, we are focusing on the creation of point clouds.

3 Generation of a photogrammetric point cloud from stereo images

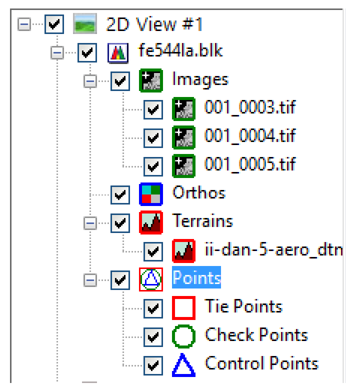
1. To generate point clouds from stereo images; first, you must open the block file. This is achieved by pressing **File → Open →** Photogrammetric Project.. . Then navigate to the block file on your computer, which is named **FE544LA.blk**:



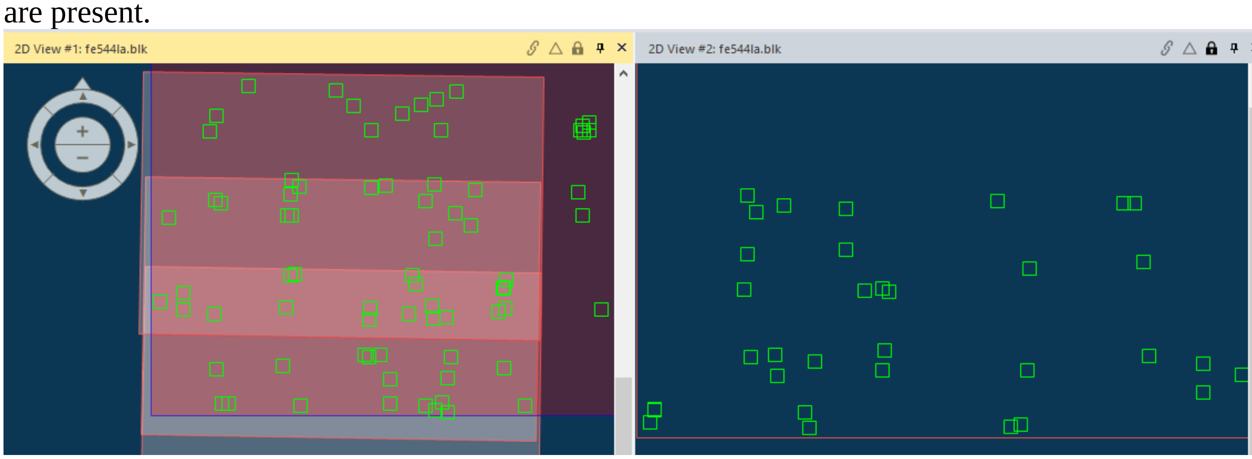
After you open the block file, the information needed to extract the 3D data is displayed in four windows: **Contents, 2D View #1, 2D View#2, and the Images.**



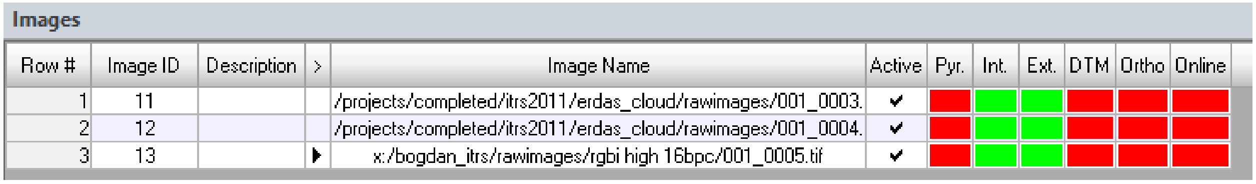
NOTE: The **Contents window** shows the main parts of the block file, namely the images, the terrain model, the tie points, the checkpoints, and the control points. The block file also contains the Orthophotos, if they are produced.



The **2D View #1** shows the footprint of the images used in the generation of the point clouds, whereas the **2D View#2** shows the points used to connect the images. As we see, only tie points are present.



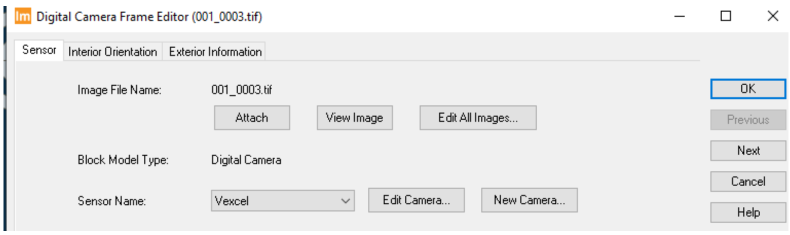
The Images window shows the image ID and whether the orientation parameters are present or not. The image ID from the block file corresponds to a physical file, which is identified by its path that is shown in the **Image Name column**. If the **Image Name** does not match the path, which is very likely given the relocation of the files between computers, the **Online column** would be red, meaning that a new synchronization of the image ID with the Image name should be done.



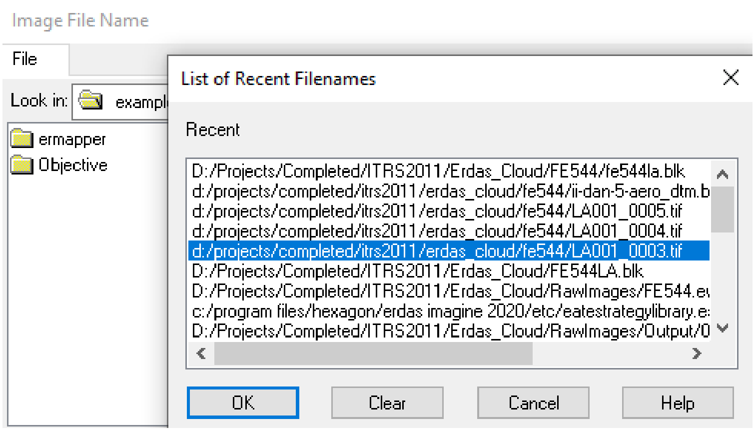
Images

Row #	Image ID	Description	>	Image Name	Active	Pyr.	Int.	Ext.	DTM	Ortho	Online
1	11			/projects/completed/itrs2011/erdas_cloud/rawimages/001_0003.	✔						
2	12			/projects/completed/itrs2011/erdas_cloud/rawimages/001_0004.	✔						
3	13		▶	x:/bogdan_itrs/rawimages/rgbi high 16bpc/001_0005.tif	✔						

2. To rebuild the relationship between the information stored inside the block file and the physical images, press the red cell under the **Online column** corresponding to each image and navigate to the correct location of the image

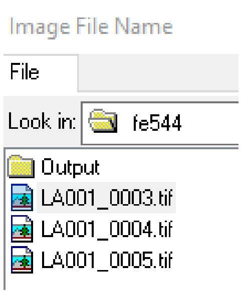


3. In the new window, click Attach to navigate to the image location:



NOTE: If you can not see the image files after you navigated to your lab folder, you may need to select the “All File-based Raster Formats option from the “Files of type” drop-down menu.

If the new folder location contains all the file from the block, they will be shown:

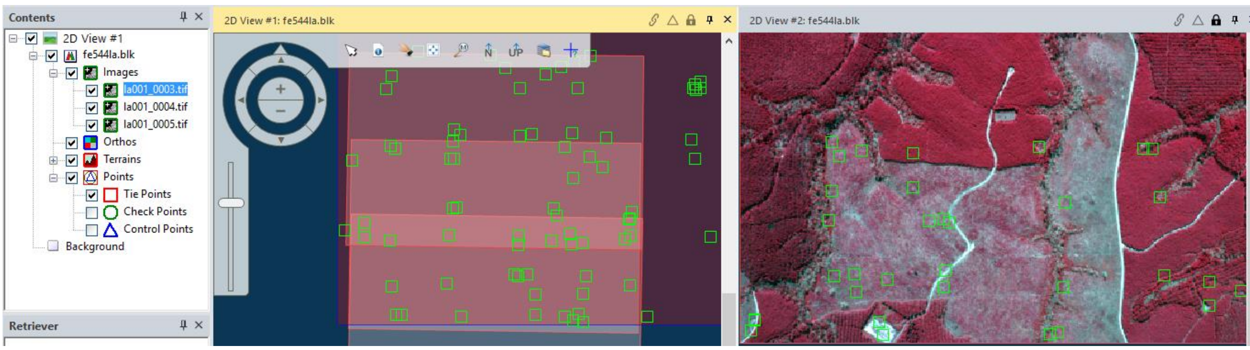


4. Choose the appropriate one, then press, OK. Once the file is added, the cells under the Online and Pyr (from Pyramid) column would turn green, which symbolizes that the file is ready to be used (if the Interior and exterior parameters are also green). Repeat the previous steps such that the rest of the tree images are ready for processing.

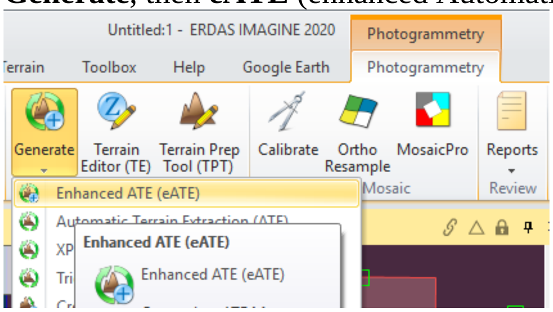
Images

Row #	Image ID	Description	>	Image Name	Active	Pyr.	Int.	Ext.	DTM	Ortho	Online
1	11			d:/projects/completed/itrs2011/erdas_cloud/fe544/la001_0003.tif	✔						
2	12			d:/projects/completed/itrs2011/erdas_cloud/fe544/la001_0004.tif	✔						
3	13		▶	d:/projects/completed/itrs2011/erdas_cloud/fe544/la001_0005.tif	✔						

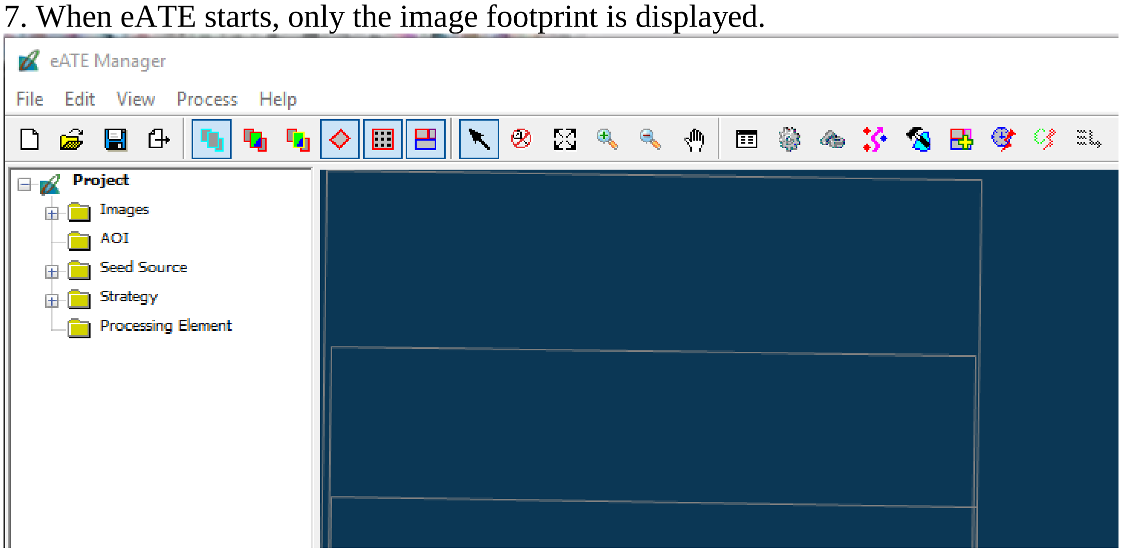
5. Once the images are properly connected within the block file, you can see them in the viewer by turning each one off then on in the **Content pane.**

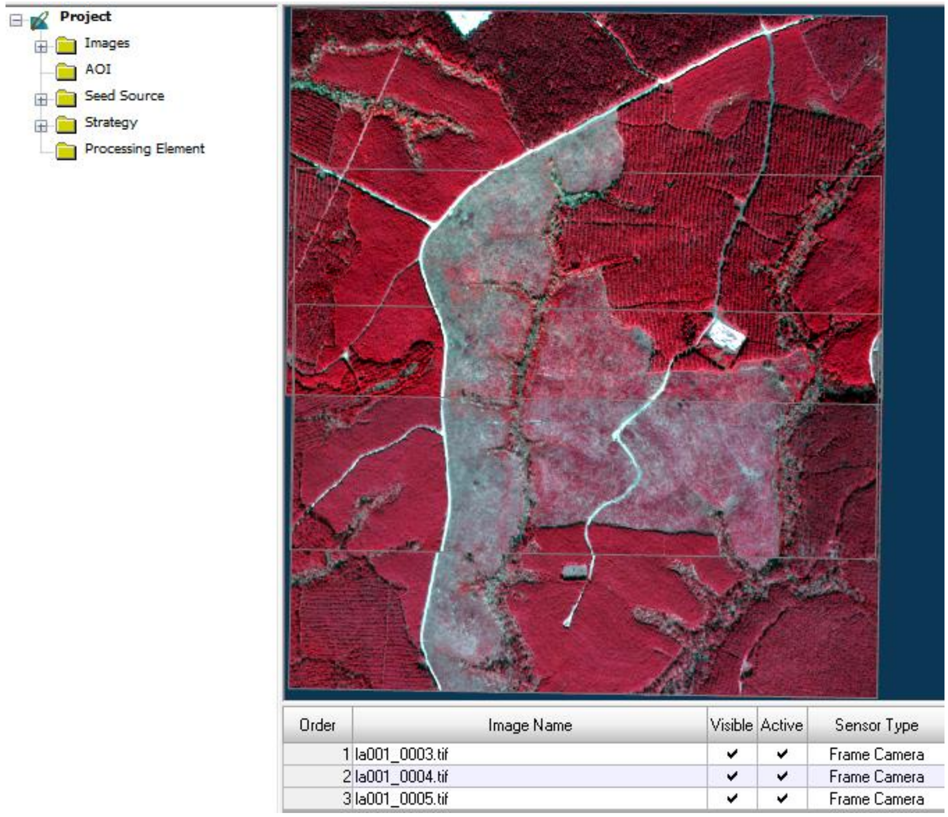


6. The point cloud generation process is executed with the **Photogrammetry tab** by selecting **Generate**, then **eATE** (enhanced Automatic Terrain Extraction).

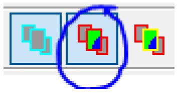


7. When eATE starts, only the image footprint is displayed.





8. To display the images you should first check the box under the **Visible column** from the table located under the viewer.

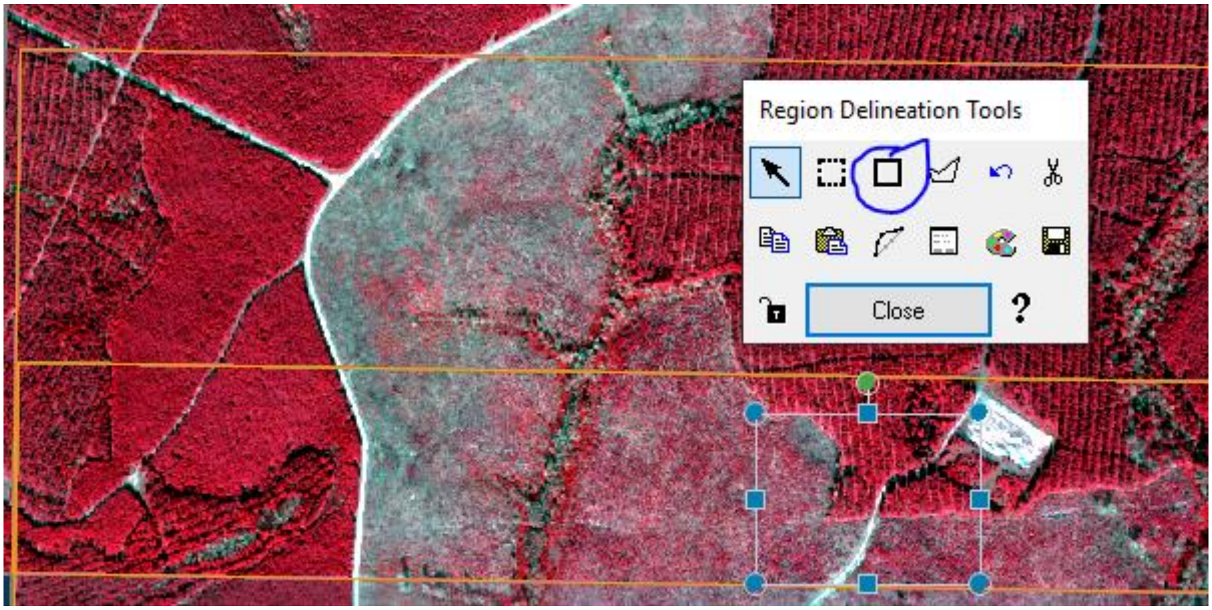
9. Next, press the **Display raster images** button from the toolbar: 

NOTE: The **Project panel** displays the information used to produce the photogrammetric point cloud: the images to be used, the area for which the point cloud will be generated, the DEM, the algorithm generating the point cloud, and the Processing element.

If you click on each folder, you will see that **Images, Seed Source** and **Strategy** have subsets, whereas **AOI** and the **Processing Element** does not. We will see later how to populate the empty options.

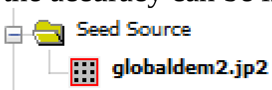
The images to be used in point cloud generation are the ones correctly positioned with the block file. The area for which the point cloud will be generated could be for the entire overlapping area or only for a smaller one, called **Area of Interest (AOI)**. If you do not select any **AOI**, the point cloud will be generated for the entire region that has overlapping images. However, In this lab, we will choose a small AOI to speed up the computation process, considering that the images are very large (more than 1 GB).

10. To choose an **AOI**, press **Start AOI Tool**,  , which will open the window called **Region Delineation Tools**. Press create rectangle **AOI icon**, then draw a rectangle on the image, like below:



After you selected the **AOI**, you should save it by pressing the **Save icon** located under **Region Delineation Tools**. After you save it, the name of the file with the AOI will open in the **AOI folder**.

The DEM provides the starting point for the extraction algorithm and can be either a general planetary surface or one that is more adjusted to the area. By default, the global DTM is used; but the accuracy can be increased is better DTM is used during the point extraction process.



The **Strategy** link shows the parameters used to generate the point clouds.
Once you are familiar with the icons on the **Project pane**, you can start the actual process of generating the photogrammetric point cloud.

To generate the point cloud, you should start by establishing the **Process Engine Settings** . The settings values have a significant impact on the processing time and the quality of the point cloud. The **Process Engine Settings window** has three tabs: **General, Format, and Radiometry.**

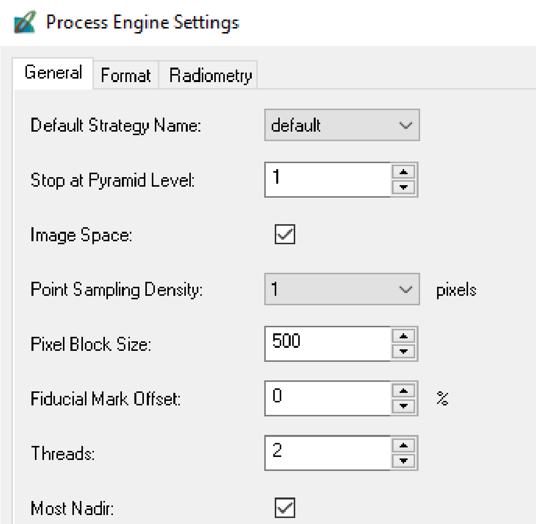
I. In the General tab, the **Stop at Pyramid Level** establishes which datasets to be used. A value of 0 means that all data would be used, which will take longer but will produce better quality point clouds. A value of 1 or 2 would be faster at the cost of accuracy. In this lab, you should choose the **value of 1.**

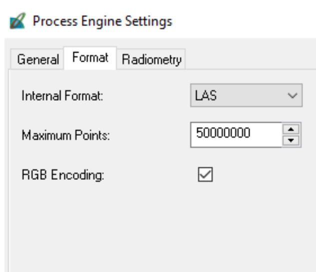
II. The **Point sampling density** decides if all pixels will be used (a value of 1) or every other pixel (a value of 2) or forth (a value of 4) etc. Smaller values lead to better results at the expense of processing time. In this lab, choose the **value of 2.**

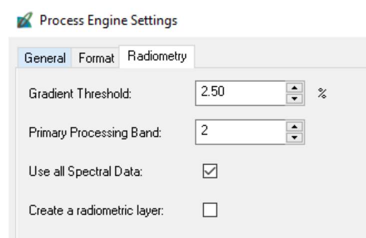
III. **Pixel Block Size** defines the number of pixels used during the processing at any moment. **Leave it at 500.**

IV. The **Format tab** hosts information on the output files, which should be left as LAS, the **maximum number of points** (which should also be left at 50 million), and whether a colored point cloud will be produced (check the **RGB Encoding**) or not.

V. The **Radiometry tab** sets up the **Gradient Threshold** if the image has poor contrast or fidelity. Increasing values will lead to more matching points but will take more time to process. **Leave it at 2.5%,** as the images have good contrast.







You can Click OK once you have finished the **Process Engine Settings**.

11.The next step in ensuring that the point cloud is generated appropriately to check the **Output Settings:**  . Similarly to the **Process Engine Settings**, there are several tabs in the **Output Settings.**

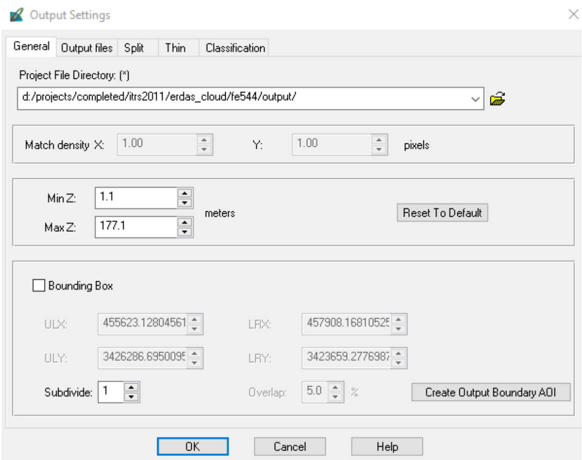
I. The **General tab** establishes where the files will be saved thru the **Project File Directory box,** so navigate to a folder that is easy to be found. If the folder does not exist, you will be asked if you wanted to create it or not. The rest of the options are self-explanatory, as they are estimated from the **block file** and the **DTM**.

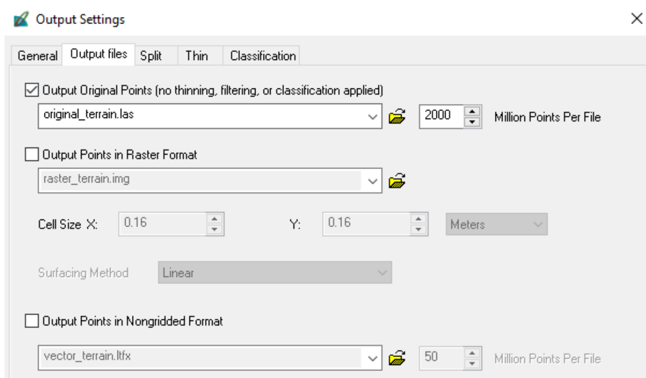
II. The **Output files** tab chooses which outputs will be produced: point clouds in a vector format, las file (**Output Original Points checkbox**), **Rater format**, or in a **Nongridded** format. You should choose the las output.

III. The **Split tab** lets you divide the output point clouds into tiles if the cloud is too big. **Do not divide the output.**

IV. The **Thin tab** lets you choose fewer points from the original point cloud if the point cloud is very large. You should **not thin the point cloud.**

V. The last tab, **Classification**, let you classify the points into various classes. Classification is very important, and we will discuss it during the lidar part of the class. For the time being, **you should not classify the point cloud.**

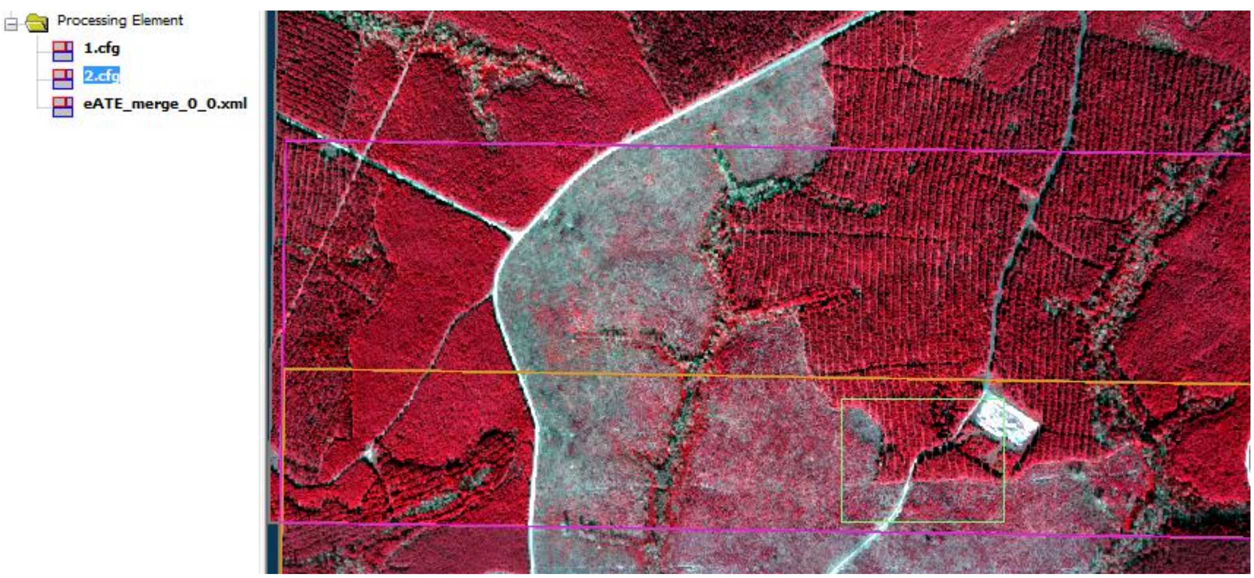




Click OK once you have chosen the appropriate settings, as described above.

The next step in the production of photogrammetric point clouds is the selection of the algorithm and its parameters that estimate the location of each point. This task is achieved in the **Strategy Manager window**. ERDAS recommends to let the options unchanged unless you are familiar with the procedures. In this lab, we will be leaving the options unchanged.

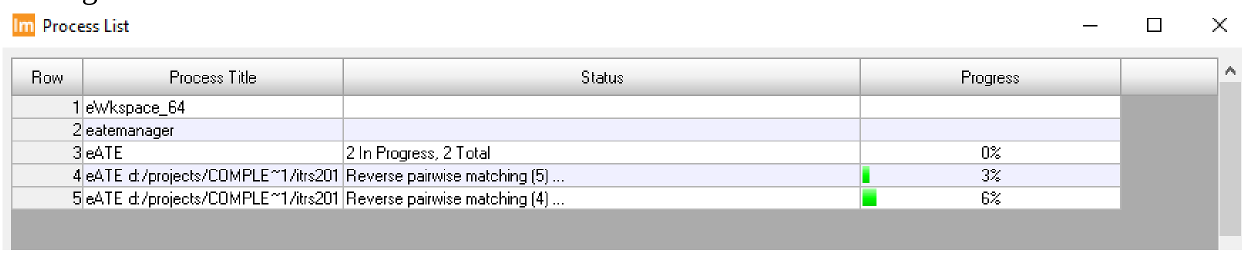
12. Once all the settings are established, press the **Generate Processing Elements button** . This will add configuration files, cfg. Each file defines the element boundaries that can be distributed across processors. To see the boundary of each element, press each configuration file. As you see, the boundary area is the overlapping area of the images.



With all the details needed to generate the point cloud from images defined, the last step is to generate the points.

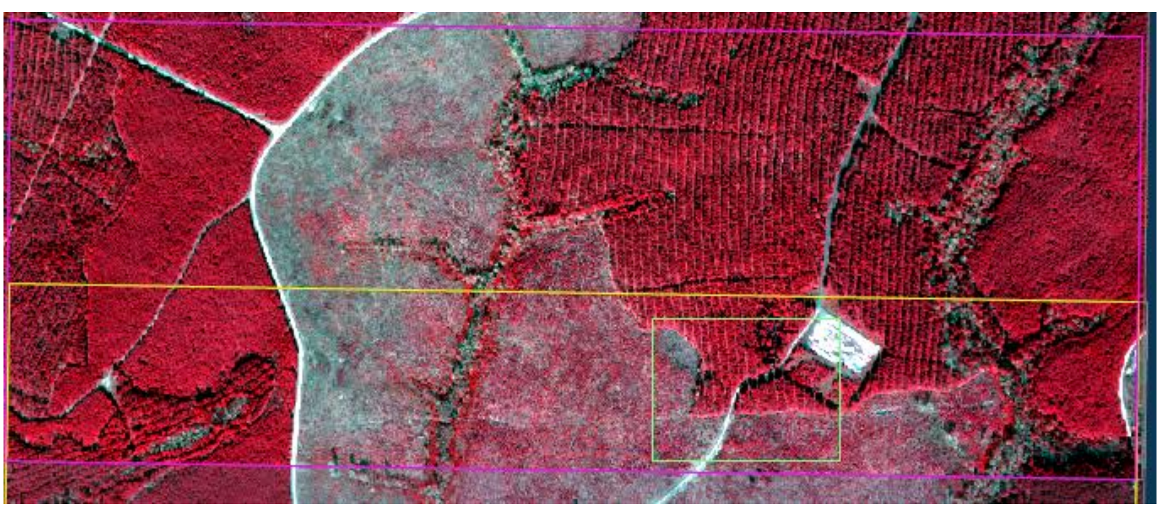
13. This task is executed by pressing the **Batch Run the eATE Process** icon: . This will open the **Batch Command Editor** that defines the computational details. You should see the two configuration files from the **Processing Element folder** listed as Inputs.
14. To finalize the generation of the photogrammetric point cloud generation, press the **Submit button**. In the **Submit window**, you have the option to start the process immediately after you **press OK** or at a later date. Choose the **Start Process Now** option.

If you have a multicore processor, which most likely is true, you can distribute the computations among CPUs; therefore, choose a number larger than 1 to speed up the process. In this case, you should choose 2 under **Simultaneous Process** because we have 2 configuration files, which means that each .cfg file will be executed on a dedicated processor. Once you press **OK,** the point cloud generation starts.



NOTE: Depending on the computation power, the image size, and the area for which the point cloud is generated, this process can take from a couple of minutes to hours.

Q3. [80%]. Generate point clouds for two areas of interest, one a small rectangle and one for the entire overlapping area of two images (for example, the purple area in the image below).



The results should contain three files:

- A Word or pdf file that has snapshots with the **Process Engine Settings Dialog windows**, namely General and Format tabs.
- Two las files.

5 Ortho-rectification

Lab developed by Rong Fang and Dr. Strimbu from ENVI and ArcGIS documentation.

5.1 Objectives:

- Orthorectify an image using RCP
- Automatically generate GCPs from an orthophoto base layer
- Assess the errors of individual GCPs and the overall model accuracy.
- Georeference an image using a vector map

5.2 Overview

An orthorectified image (or orthophoto) is one where each pixel represents a true ground location, and all geometric, terrain, and sensor distortions have been removed to within a specified accuracy. Orthorectification transforms the central perspective of an aerial photograph or satellite-derived image to an orthogonal view of the ground, which removes the effects of sensor tilt and terrain relief. Scale is constant throughout the orthophoto, regardless of elevation, thus providing accurate measurements of distance and direction. Geospatial professionals can easily combine orthophotos with other spatial data in a geographic information system (GIS) for city planning, resource management, and other related fields.

An RPC model is a type of model that maps the physical relationship between 3D ground points and 2D image points. RPCs are not the same as a map projection. Most modern high-resolution sensors such as WorldView-3 and Pleiades include pre-computed RPCs with their imagery. The ENVI RPC Orth rectification tools use RPCs—along with x/y/z data from GCPs and elevation data from a high-resolution DEM—to create an orthorectified image.

Automatic GCP generation is based on image matching between the reference and source images, so the scene contents should not be vastly different. Automatic GCP generation is more robust if the reference image and input image have a similar resolution. If the reference image has a much higher resolution than the input image (i.e., the ratio is greater than 2.5), down-sampling the reference image should be executed first. The DEM must cover at least the spatial extent of the input raster.

5.3 Description of the files

- **OrbViewSubset.dat** is a spatial subset of an OrbView-3 panchromatic image of Coral Springs, Florida, acquired on 16 January 2006. The image has the RPC information embedded in the hdr file.
- **NAIPReferenceImage.dat** is a spatial subset of a National Agriculture Imagery Program (NAIP) image, acquired on 07 August 2007.
- **DEM.dat** is a spatial subset of a National Elevation Dataset (NED) DEM image at 1/9 arc-second resolution.
- **BolderSPOT.dat** is a SPOT image of Boulder, CO from 1998.
- **Roads** is a vector format of some roads from Boulder, CO, which are stored as a shapefile. The shapefile is made up of at least four files: .shp, .shx, .dbf, and .prj.

5.4 Orthorectification using RPC and DEM

Most images supplied by an airborne or spaceborne sensor are unrectified, therefore there is distortion across the image caused by the terrain. The orthorectification process of an image removes the distortions and creates a planimetric image at every location (orthorectified) with a consistent scale everywhere within the image. In summary, orthorectification applies a transformation (stretches) to the image to match the spatial accuracy of a map by considering location, elevation, and sensor information.

An accurately orthorectified image can be produced using the rational polynomial coefficients (RPCs) in conjunction with an accurate digital elevation model (DEM). Many companies, such as Digital Globe or GeoEye, deliver some of their images with the RPCs, which can be used to geometrically correct the unrectified image. DigitalGlobe stores RPC information in a .rpb file, whereas GeoEye images have the RPC information stored in a rpc.txt file. ArcGIS automatically reads RPC information in the associated text file (if it can be found next to the image file) and applies the RPC transformation (image correction) on the fly when displaying it in ArcMap. However, RPC information can be stored in the files defining the image, which is the case of ENVI images. The dat files used by ENVI store the RPC information inside the header file, hdr, which is in essence, an ASCII file.

Q1 [10%]. Why orthorectification is important (at most three sentences)?

Q2 [10%]. What is the main difference between parametric and nonparametric orthorectification (at most two sentences)?

Q3 [10%]. Are RPCs popular in airborne or spaceborne orthorectification (at most two sentences)?

Lister - [c:\Users\bogda\Box\Courses\FE444\FE444F2020\Week05\FE444w05_Data\OrbViewSubset.hdr]

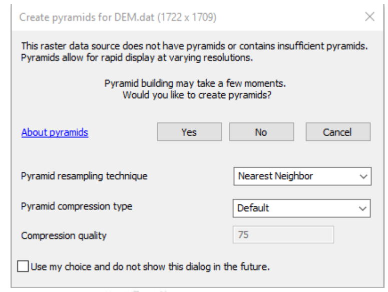
File Edit Options Encoding Help

```
ENVI
samples = 4905
lines   = 5364
bands   = 1
data type = 12
interleave = bsq
file type = ENVI Standard
header offset = 0
byte order = 0
rpc info = {
  1.48480000e+004,  4.00800000e+003,  2.62445000e+001, -8.02661000e+001,
  3.00000000e+000,  1.48480000e+004,  4.00800000e+003,  1.35300000e-001,
  4.22000000e-002,  2.99000000e+002, -1.90335570e-003,  3.42466780e-003,
 -1.01582290e+000,  1.60480660e-003, -4.44856530e-003,  1.28617840e-005,
 -1.39847330e-003, -3.61894430e-005,  1.47050170e-002,  2.09288310e-006,
  6.74111760e-004, -5.50363340e-006, -4.31784780e-003, -1.04302460e-006,
  1.53600800e-003,  5.51892060e-003,  3.37075120e-010, -4.78163540e-006,
 -7.79155000e-004,  5.50255390e-009,  9.99902160e-001,  8.02944130e-003,
 -8.22848630e-003,  2.58220320e-003,  4.47673770e-003, -6.44857160e-004,
  7.24541610e-004, -1.48810650e-003, -3.94912810e-003,  2.83260900e-006,
  3.94803060e-006, -5.70953980e-006, -1.27577960e-005, -8.00780350e-007,
  6.99661310e-006,  2.07574750e-005,  9.47323550e-007, -4.31589860e-006,
 -4.70985470e-006,  4.22293260e-009, -1.16385970e-003,  1.03184680e+000,
  4.61249990e-004, -1.38569600e-002,  9.00087040e-003,  1.63542920e-003,
 -1.56559950e-003,  4.87829170e-003,  2.88429480e-004, -1.99504460e-005,
  1.74844760e-004,  7.32830820e-005,  3.49213060e-003,  1.10243670e-005,
  1.55654010e-003, -4.40233340e-006, -3.16561230e-006,  9.09044960e-005,
 -5.35696880e-005, -1.91039220e-007,  9.99546570e-001,  2.93979230e-003,
  2.15719590e-002,  9.45285410e-004,  1.30807910e-003,  8.33854610e-005,
  1.80445580e-004,  6.71460070e-005,  5.64288600e-003,  1.07976450e-005,
 -2.13915830e-007, -3.42033130e-007, -1.05471650e-006, -8.38575640e-008,
 -1.15735720e-006, -5.79459240e-006,  1.40185140e-007, -1.97597060e-007,
 -9.60895720e-007, -8.00861270e-009,  8.15600000e+003,  9.10000000e+002,
  1.00000000e+000}
x start = 911
y start = 8157
wavelength = {
 0.625000}
fwhm = {
 0.450000}
wavelength units = Micrometers
data gain values = {
 2.34490000e-001}
data offset values = {
 0.00000000e+000}
sun azimuth = 153.110000
sun elevation = 37.980000
solar irradiance = {
 1.49300000e+003}
cloud cover = 999.000000
sensor type = OrbView-3
acquisition time = 2006-01-16T16:01:07.904400Z
```

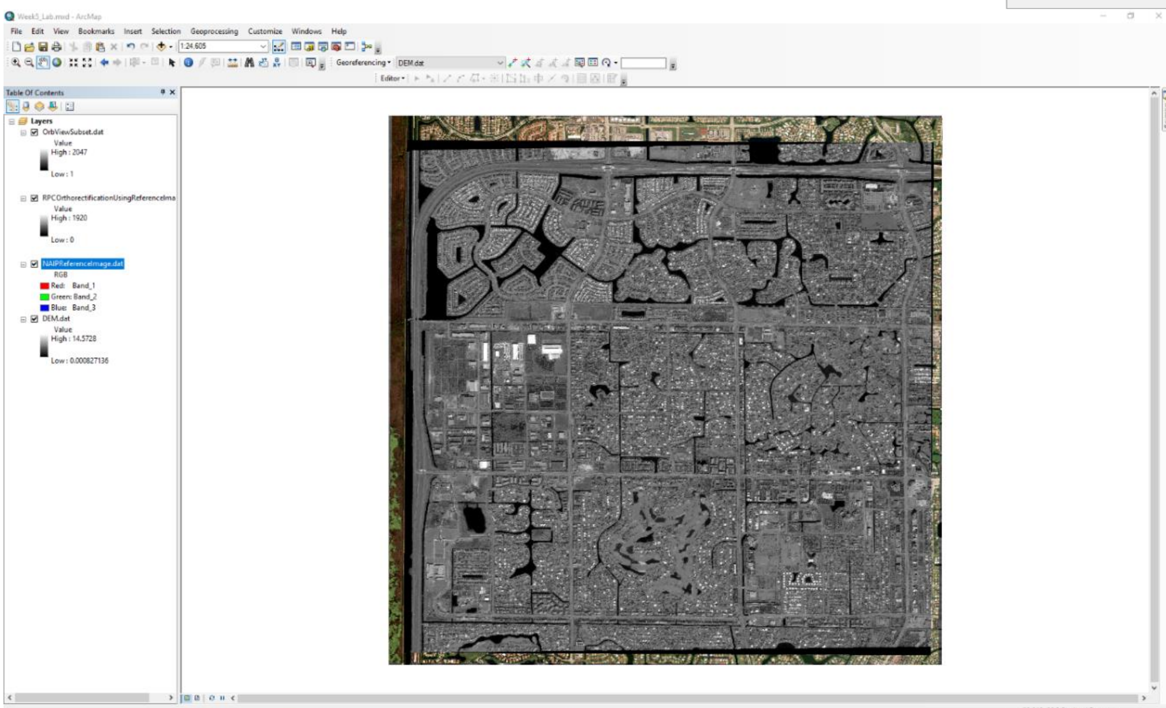
As you see, there is no coordinate system inside the header file, just the RPC info, which means that the image needs to be orthorectified and georeferenced. Usually, this process happens simultaneously in almost all the packages used for orthorectification. Therefore, the tools used for orthorectification will include the georeferencing tools too.

Launch ArcGIS, and choose a blank ArcMap page. Do not forget to save the file often, such that you will not lose a lot of your work in eventuality that ArcGIS or the PC will crash.

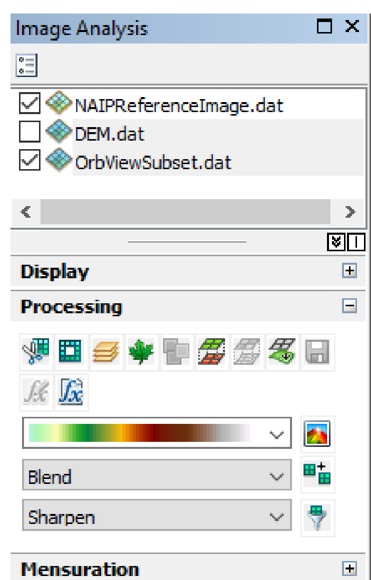
1. In the main menu, click the **Open button** .
2. Navigate to the directory where you saved the Lab data. Use **Ctrl-click** to select the **OrbViewSubset.dat, NAIPReferenceImage.dat, and the DEM.dat**
3. Click yes for creating pyramids.



The image will be displayed as follow:



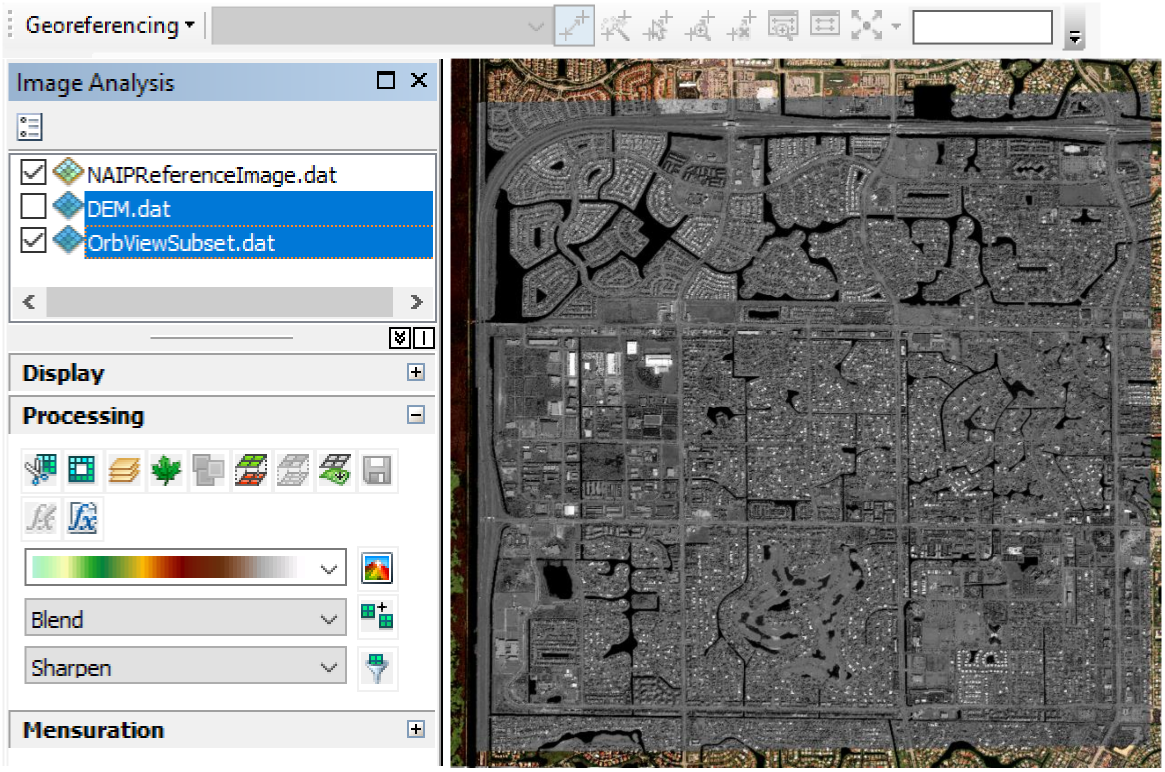
To start the orthorectification process, you should turn on the interface responsible for orthorectification, which is accessed by choosing **Windows** -> **Image Analysis.** This will open the Image Analysis windows.



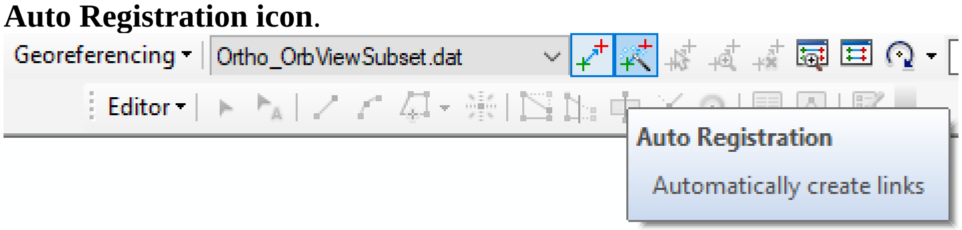
1. Under image analysis, select **OrbViewSubset.dat** and **DEM.dat.**

   Click **Orthorectify** button: . A temporary orthoimage is created **(Ortho_OrbViewSubset.dat**), which is shown on the top of the **Table of Contents**. The orthoimage is overlapped on the top of the reference image with a roughly similar location.

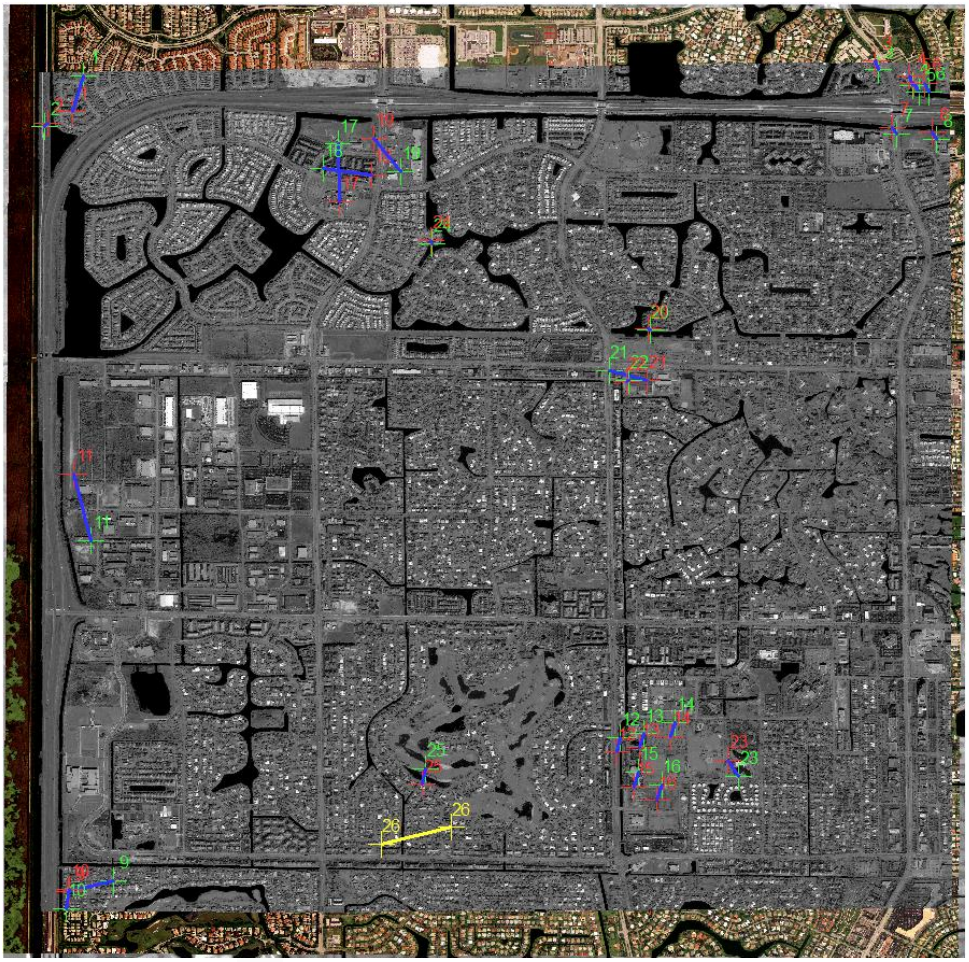
2. In the main menu, choose **Customize → Toolbars → Georeferencing**. This will display the georeferencing toolbar.



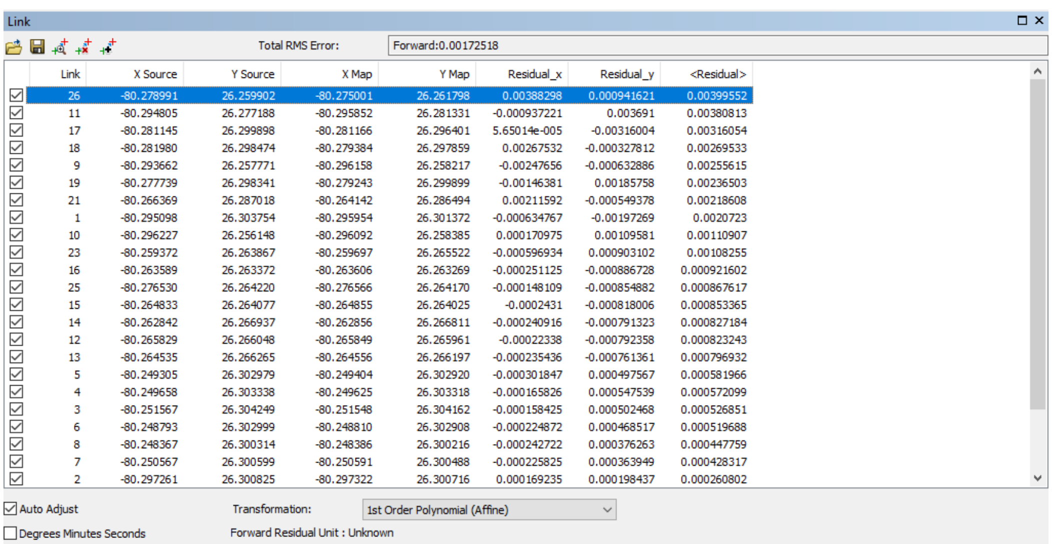
3. After the image was orthorectified, it should be accurately georeferenced. In this lab, we will georeference it using another image, namely the NAIP imagery. In the Image, Analysis windows check the **Ortho_OrbViewSubset.dat** and **NAIPRefereneImage.dat**. In the Georeferencing toolbar, select **Ortho_OrbViewSubset** active image, and click on , the **Auto Registration icon**.



4. Auto registration will generate a series of **ground control points (GCPs)** to link the two images.



5. Click on **View Link Table** in the georeferencing toolbar; you will see the summary of link table.



Link

Total RMS Error: Forward:0.00172518

Link	X Source	Y Source	X Map	Y Map	Residual_x	Residual_y	<Residual>
26	-80.278991	26.259902	-80.275001	26.261798	0.00388298	0.000941621	0.00399552
11	-80.294805	26.277188	-80.295852	26.281331	-0.000937221	0.003691	0.00380813
17	-80.281145	26.299898	-80.281166	26.296401	5.65014e-005	-0.00316004	0.00316054
18	-80.281980	26.298474	-80.279384	26.297859	0.00267532	-0.000327812	0.00269533
9	-80.293662	26.257771	-80.296158	26.258217	-0.00247656	-0.000632886	0.00255615
19	-80.277739	26.298341	-80.279243	26.299899	-0.00146381	0.00185758	0.00236503
21	-80.266369	26.287018	-80.264142	26.286494	0.00211592	-0.000549378	0.00218608
1	-80.295098	26.303754	-80.295954	26.301372	-0.000634767	-0.00197269	0.0020723
10	-80.296227	26.256148	-80.296092	26.258385	0.000170975	0.00109581	0.00110907
23	-80.259372	26.263867	-80.259697	26.265522	-0.000596934	0.000903102	0.00108255
16	-80.263589	26.263372	-80.263606	26.263269	-0.000251125	-0.000886728	0.000921602
25	-80.276530	26.264220	-80.276566	26.264170	-0.000148109	-0.000854882	0.000867617
15	-80.264833	26.264077	-80.264855	26.264025	-0.0002431	-0.000818006	0.000853365
14	-80.262842	26.266937	-80.262856	26.266811	-0.000240916	-0.000791323	0.000827184
12	-80.265829	26.266048	-80.265849	26.265961	-0.00022338	-0.000792358	0.000823243
13	-80.264535	26.266265	-80.264556	26.266197	-0.000235436	-0.000761361	0.000796932
5	-80.249305	26.302979	-80.249404	26.302920	-0.000301847	0.000497567	0.000581966
4	-80.249658	26.303338	-80.249625	26.303318	-0.000165826	0.000547539	0.000572099
3	-80.251567	26.304249	-80.251548	26.304162	-0.000158425	0.000502468	0.000526851
6	-80.248793	26.302999	-80.248810	26.302908	-0.000224872	0.000468517	0.000519688
8	-80.248367	26.300314	-80.248386	26.300216	-0.000242722	0.000376263	0.000447759
7	-80.250567	26.300599	-80.250591	26.300488	-0.000225825	0.000363949	0.000428317
2	-80.297261	26.300825	-80.297322	26.300716	0.000169235	0.000198437	0.000260802

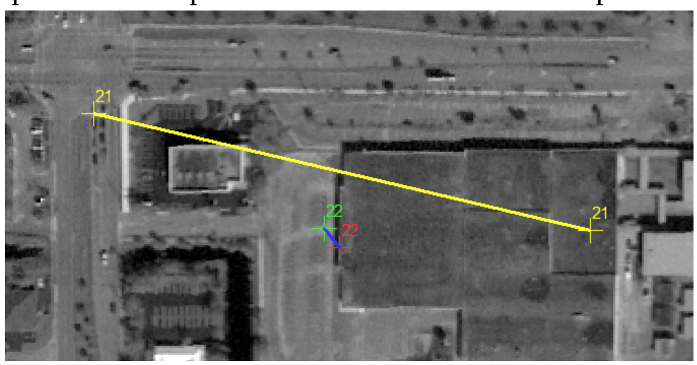
Auto Adjust   Transformation: 1st Order Polynomial (Affine)

Degrees Minutes Seconds   Forward Residual Unit : Unknown

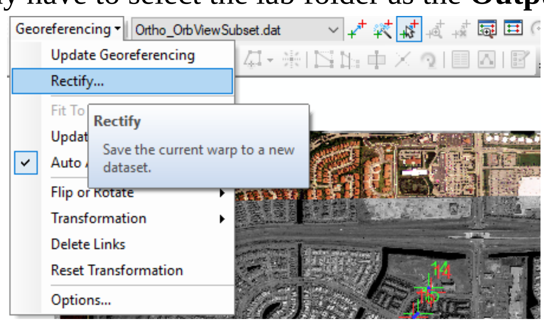
6. You can examine the total Root Mean Square Error (RMSE). Display the links by descending order, to reduce the total RMS by deleting some of the links. To delete links with big error, select the link and click on the **Delete Link** icon .

Q4 [10%]. What is the GCP with the smallest Error Magnitude value? And what is your total RMS Error?

7. Exam the GCPs on the images. If you find the same GCP link is distant in the image, you can adjust their position to be as close as possible or delete it. For example, the pair of point 21 is far from each other, and you can move the point by using the **Select Link** icon (in georeferencing toolbar) , or delete them. After the adjustment, reexamine the table of links. Repeat this step until all the pairs of links are within an acceptable distance.



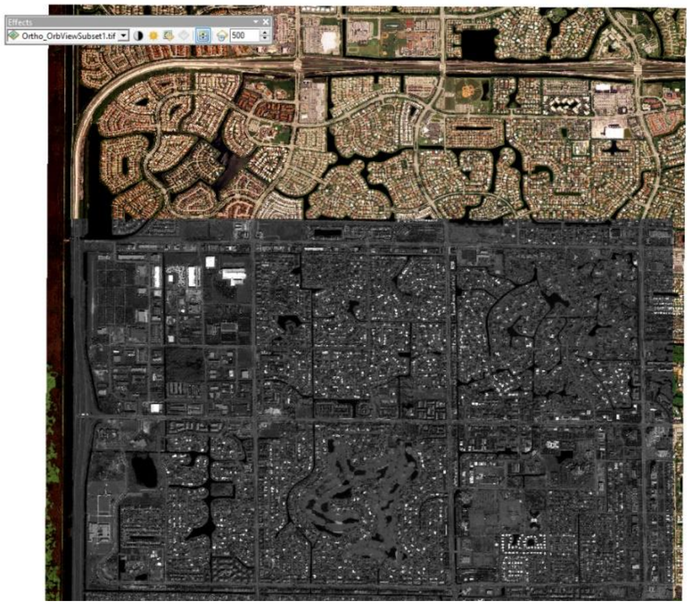
8. Save the link after you finish correcting the error. Open the table of link by click on . And in the table of link, click on , and save the .txt file under the folder of Week 5. Name it **GCPs_Lab5.**
9. Rectify and create a new image file. After you finish correcting the links, select **Rectify** under **Georeferencing** (You may have to select the lab folder as the **Output location).**



Leave all the settings as default. Save the image as **Ortho_OrbViewSubset1.tiff.**

10. Click on Add the **Ortho_OrbViewSubset1** layer to ArcMap.
11. Compare **Ortho_OrbViewSubset1** with **NAIPReferenceImage**. In the main menu, click on **Customize →Toolbars,** select **Effects.** Turn on the Effects toolbar. Click on to swipe the image for comparison.

Q5 [10%]. Include snapshots of view with all the links (GCPs) distributed and Views of the image Ortho_OrbViewSubset1.



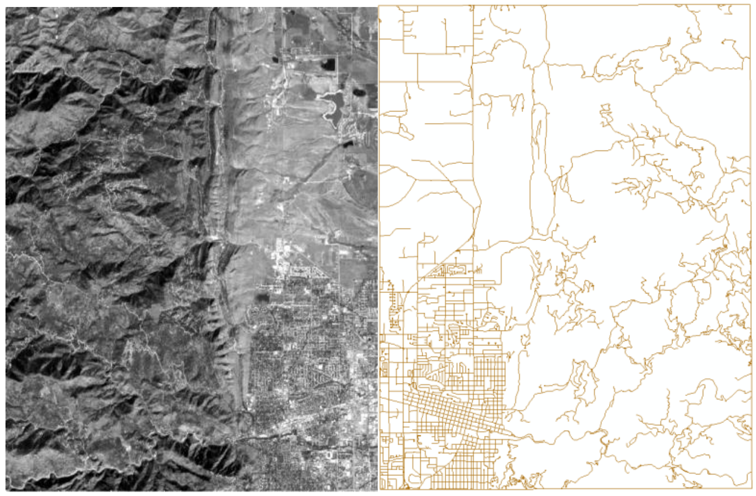
5.5 Georeferencing an image using a map (vector)

In many instances, the image is orthorectified but lacks the coordinate system. Therefore, you have to position the image in the correct location and to scale properly the image. When georeferencing using a vector layer, you should identify well-defined objects within your images, such as road intersections or land features, that can be found also in the vector. The selection of obvious land features ensures that the same locations in both the raster and the vector layer(s) are referenced.

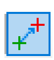
The data used for this exercise is the SPOT image for the city of Boulder, CO, from 1998, and the road network from the same area is stored as a shapefile (Roads.shp). The SPOT image does not have coordinates, and therefore must be georeferenced. To georeference an image in ArcGIS, we will be using the georeferencing tool that was used before.

To display the **Georeferencing toolbar**, click the **Customize menu** and **click Toolbars > Georeferencing.**

1. Upload the two files in ArcGIS: **BoulderSPOT.dat and Roads.shp.**



2. From the **Georeferencing toolbar**, click the drop-down arrow and select **BolderSPOT.dat** if its not already selected.

3. Click **Fit To Display (under Georeferencing drop-down menu)**, which will show the image in the same area as the target layers. You can Shift and Rotate the raster as needed. To see all the datasets, adjust their order in the table of contents.
4. To georeferenced the image, you will use the same procedure as in the previous exercise, namely, find points that are visible in both files, then match them. Therefore, Click the Add Control Points tool  to add control points. To add a link, first, you should click a known location on the raster dataset, then click a known location on the vector layers in the Roads map, which serves as the reference data. You should be aware that the Rotate and Shift tools are not available after you place the first link.
5. Add at least 10 links to georeference the image. Remember that you need a minimum 3 links for a first-order polynomial, 4 links for a projective transformation, 6 links for a second-order polynomial, and 10 links for a third-order polynomial or spline transformation. You should notice that as soon as you add points, the image starts being moved to its possible location, making the selection of points easier.

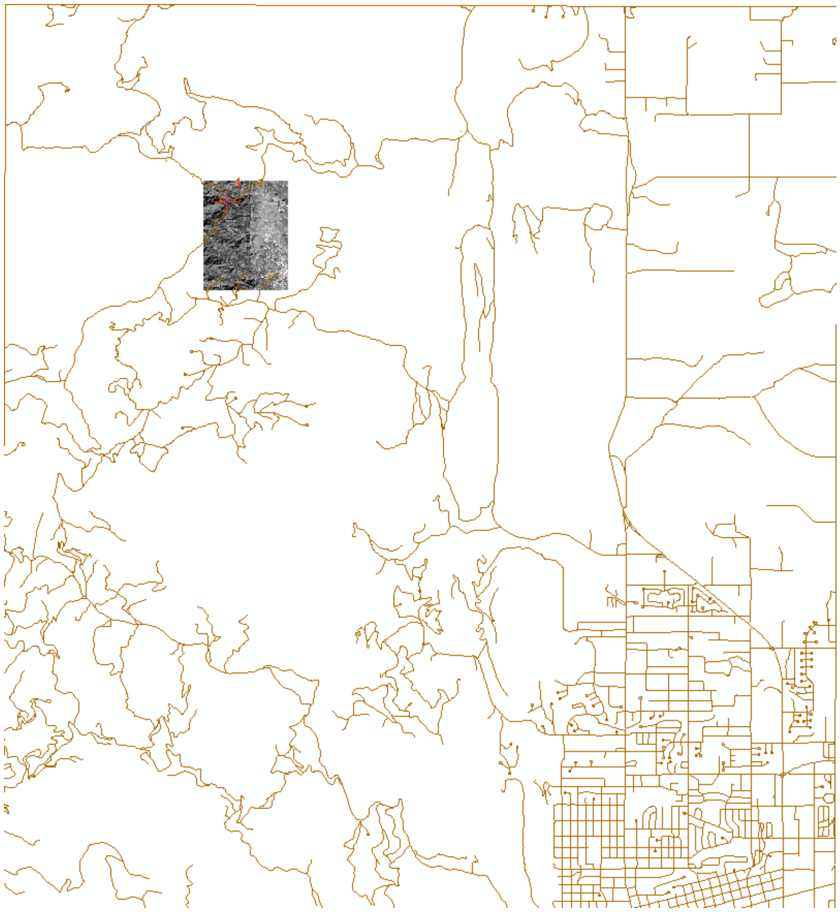


Image located after the first link

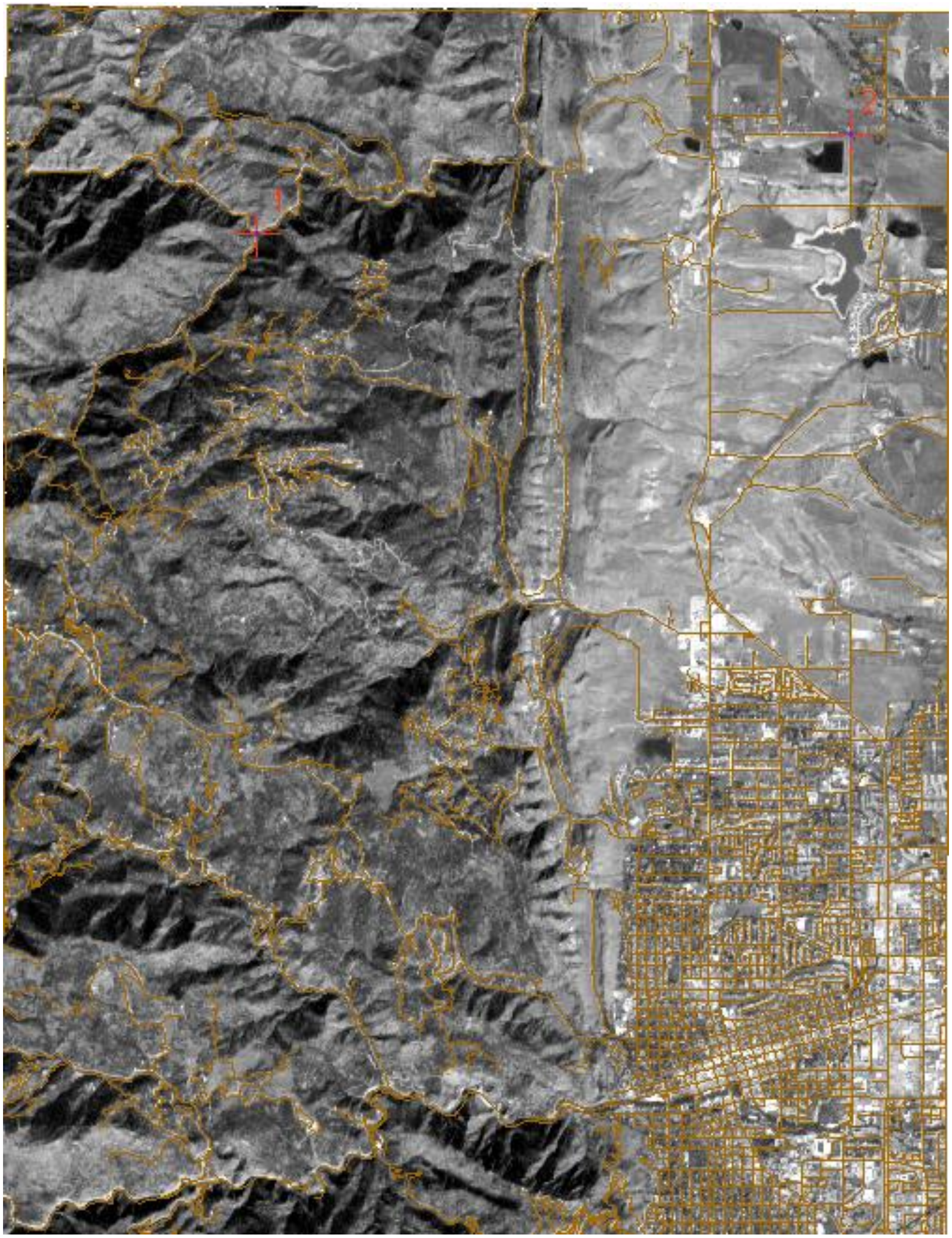
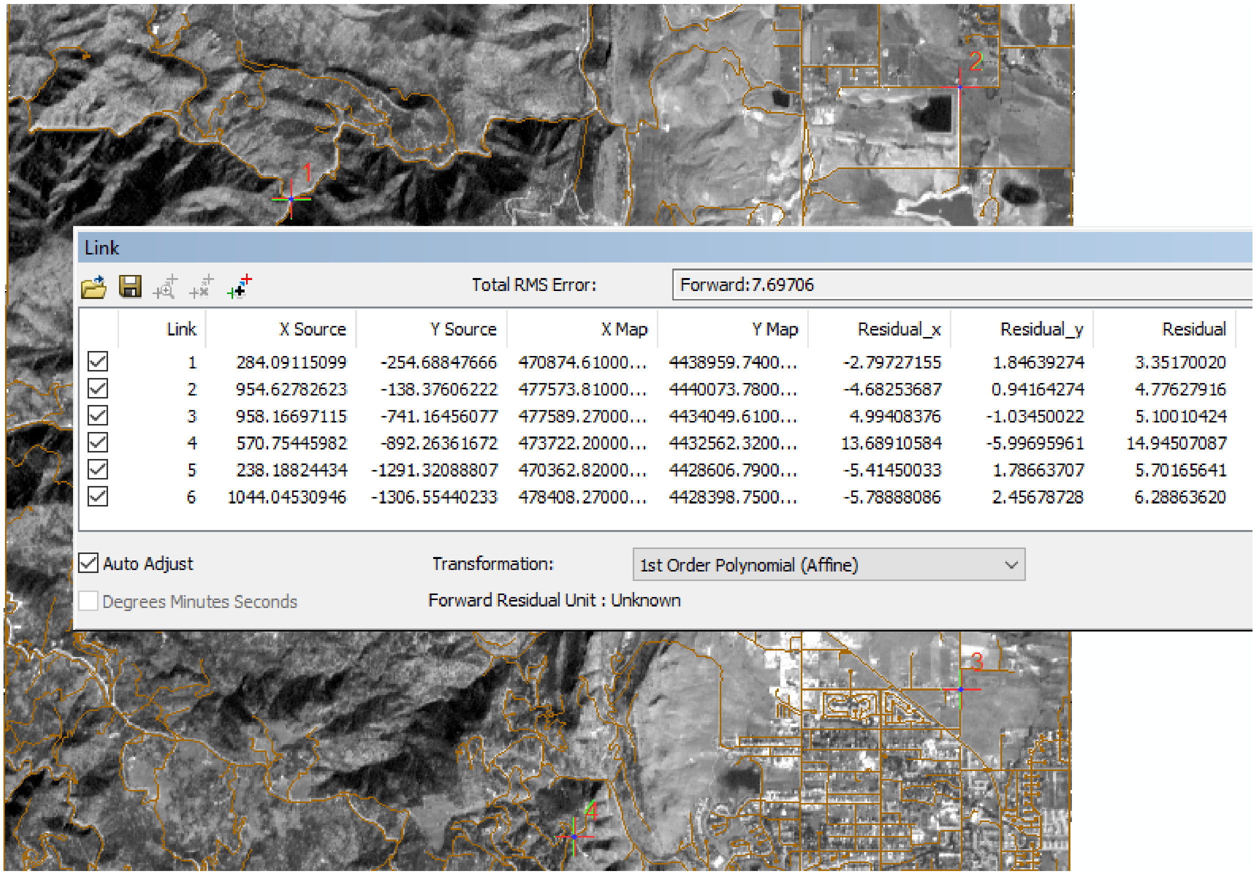


Image located after the second link

6. To evaluate the transformation, click the **View Link Table** button , where you can examine the RMSE for each link. If you're satisfied with the registration, meaning a RMSE<1, then you can stop entering links. To adjust the transformation, you can delete any unwanted links from the Link table. Remember that selection of the points is a tedious process that requires patience, as many points should be selected, deleted, or adjusted.



Link table for the first 6 links.

7. Once you reach a comfort level in respect with the RMSE and number of links, click the **Georeferencing drop-down** menu and click either **Update Georeferencing or Rectify**. Updating the georeferencing will save the transformation information with the image as well as its auxiliary files, whereas Rectifying will create a new file with the georeferencing information.

Q6 [10%]. Identify the links with the largest and smallest RMSE. Provide a snapshot with the Link table.

Q7. [10%] Why there is a need of more than 3 links for an affine transformation (first-degree polynomial)?

Q8.[10%]. Execute a first, second, and third-order polynomial transformation. What do you notice that happens to the image? Answer with at most five sentences.

Q9. [10%]. Why you are recommended to spread the links (you can think on them as GCP) thought-out the image rather than on corners or on a line across the image? Provide a short answer, no longer than 3 sentences.

Q10. [10%]. Include snapshots with all the steps needed for georeferencing.

5.6 Advanced topics: Ortho-rectification and Georeferencing

Lab developed by Sudeera Wickramarathna and Dr. Strimbu from ERDAS documentation.

5.6.1 Objectives:

- Learn two different methods for performing rational polynomial coefficients (RPC) orthorectification
- Automatically generate GCPs from an orthophoto base layer,
- Use GCPs interactively to orthorectify the image
- Assess the errors of individual GCPs and the overall model accuracy.
- Automatically generate GCPs and perform RPC orthorectification in one step
- Extract Ground Control Points (GCPs) from a vector layer (of roads)
- Image to map registration:
    - build a warp model and georegister a SPOT scene to the map projection of the vector roads layer
    - use RMS error values to fine tune and adjust your warp model
- use a vector layer (in this case the roads) to assess the accuracy of the registration results
- automatic registration of image to image

5.6.2 Overview

An orthorectified image (or orthophoto) is one where each pixel represents a true ground location, and all geometric, terrain, and sensor distortions have been removed to within a specified accuracy. Orthorectification transforms the central perspective of an aerial photograph or satellite-derived image to an orthogonal view of the ground, which removes the effects of sensor tilt and terrain relief. Scale is constant throughout the orthophoto, regardless of elevation, thus providing accurate measurements of distance and direction. Geospatial professionals can easily combine orthophotos with other spatial data in a geographic information system (GIS) for city planning, resource management, and other related fields.

An RPC model is a type of model that maps the physical relationship between 3D ground points and 2D image points. RPCs are not the same as a map projection. Most modern high-resolution sensors such as WorldView-3 and Pleiades include pre-computed RPCs with their imagery.The ERDAS RPC Orthorectification tools use RPCs—along with x/y/z data from GCPs and elevation data from a high-resolution DEM—to create an orthorectified image.

Automatic GCP generation is based on image matching between the reference and source images, so the scene contents should not be vastly different. Automatic GCP generation is more robust if the reference image and input image have a similar resolution. If the reference image has a much higher resolution than the input image (i.e., the ratio is greater than 2.5), down-sampling the reference image should be executed first. The DEM must cover at least the spatial extent of the input raster.

In many applications, it is critical to have an image align with either another image or a map. For instance, one major application where alignment is critical is image change detection. If you wanted to compare two images from different dates to see where change had occurred, unless those two images matched up well (in terms of the geographic positions that they represent), accurate change detection would be impossible. In order to ensure that images match up with each other, they must be represented by the same coordinate/reference system. The process of Image georectification/ georegistration involves associating a remotely sensed image with a particular coordinate/ reference system. There are many ways that this can be done, two such approaches being image to map registration and image to image registration.

5.6.3 Description of the files

- **OrbViewSubset.dat** is a spatial subset of an OrbView-3 panchromatic image of Coral Springs, Florida, acquired on 16 January 2006.
- **NAIPReferenceImage.dat** is a spatial subset of a National Agriculture Imagery Program (NAIP) image, acquired on 07 August 2007.
- **DEM.dat** is a spatial subset of a National Elevation Dataset (NED) DEM image at 1/9 arc-second resolution.
- BolderSPOT.dat is a SPOT image of Boulder CO from 1998
- Roads.evf is a vector format of some roads from Boulder CO.
- ikonos_4.0m.dat is IKONOS true-color image of Boulder, CO, at 4 meter spatial resolution
- ikonos_4.0m.hdr is the header file of ikonos_4.0m.dat
- quickbird_2.4m.dat is QuickBird true-color image of Boulder, CO, at 2.4 meter spatial resolution
- quickbird_2.4m.hdr is the header of the quickbird_2.4m.dat

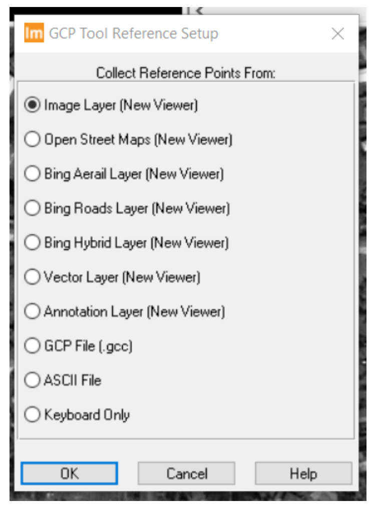
5.6.4 Rectification Workflow

1. In the **File Tab**, click the Open button, and then Raster Layer option in **2DView #1**.
2. Navigate to the Lab 5 directory where you saved the Lab data. Select and open **OrbViewSubset.hdr** file.

NOTE: If you cannot find the respective images/files after navigating the Lab folder, you may need to select **All File-Based Raster Formats** from the Files of Types drop-down list.
If you want to make the current directory as the default directory, you can click icon (third icon from left). Also, you can select a default output folder by clicking icon (first icon from the right). These options will help you to select and add the files quickly to the ERDAS Content pane.

3. Now right click on the image and select **Fit to Frame** to see the entire image. Then click on the **Metadata** option in the **Home Tab** so that you can view the image metadata.
4. Once the **Image Metadata** dialog box is open, check the Projection Info: You will notice this image is missing spatial reference information.
   In the next steps, we are going to add spatial information to this image with the rectification process. Check if any other ERDAS windows opened. If so, close them.
5. Go to **Raster|Panchromatic Tab** and click **Control Points** option 

 under **Transform and Orthocorrect** group.
6. Next, **Set Geometric Model** dialog box will open, select **Model List** option, and scroll down the **Select Geometric Model** list. Select the **Polynomial model** from the list.



7. **GCP Tool Reference Setup** dialog box will open and select the **Image Layer** option and click **OK**.

8. Now **Reference Image Layer** will open, and you can enter the image that we are going to get the geographic reference points. Select "**naipreferenceimage.img**" as the reference image.

9. The **Reference Map Information** dialog box will open with the **Reference information**. Check the information and click **OK.**

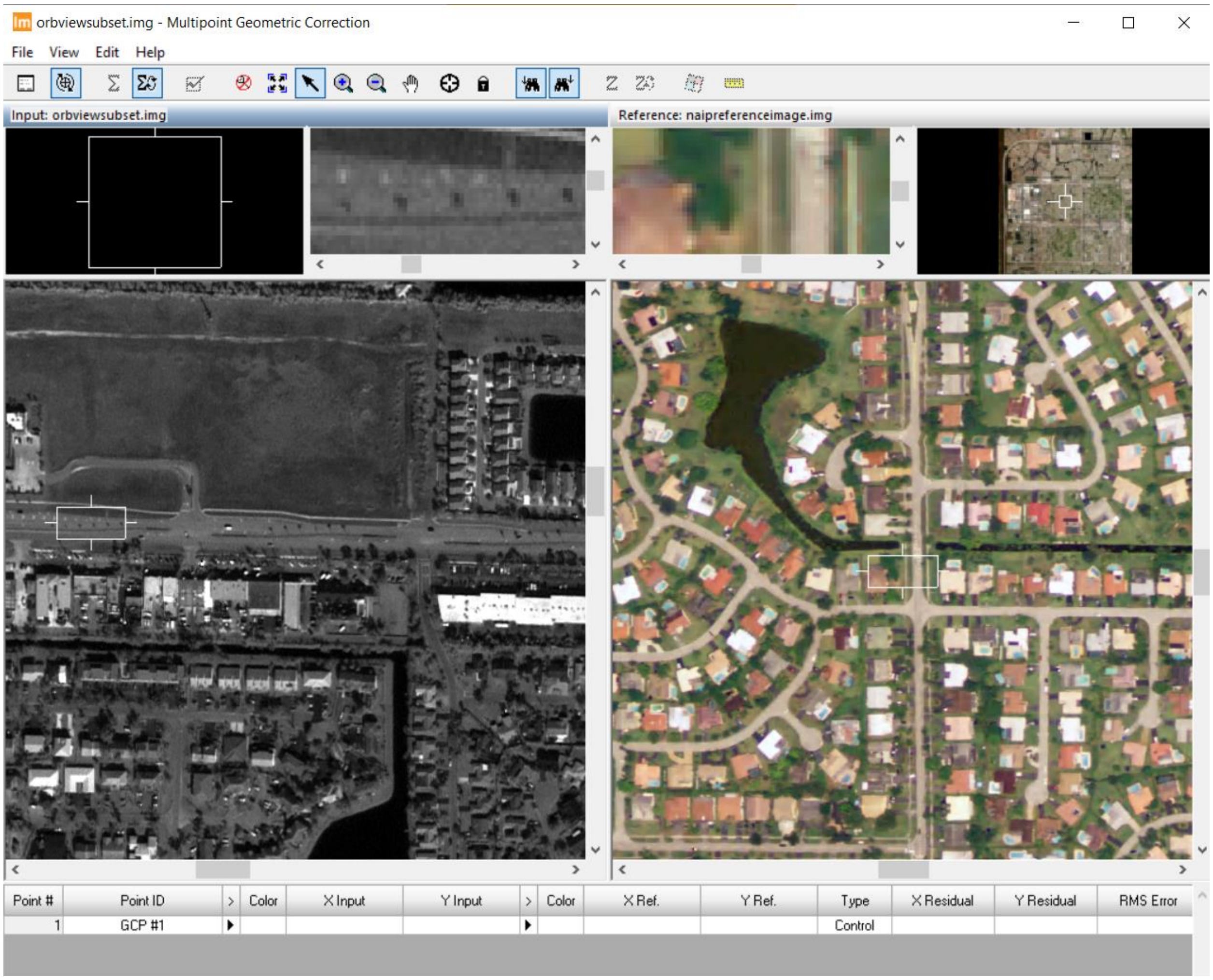
10. Then **Polynomial Model Properties** Box will appear. You can select the **Polynomial Order as 1** by using the up-down arrows.

NOTE: If the **Apply** button is not activated yet, you can change the Polynomial Order to 2 and then the down arrow to return it to 1. This will allow you to be active in the Apply option.

11. Click **Apply** and close the dialog box. Now, you should get a separate window (**Multipoint Geometric Correction (MGC)**) as follows:

NOTE: This MGC window has two major panels. The right-hand side panel has the NAIP image, which is the reference image with geographic reference information. On the left-hand side panel, we have the uncorrected aerial image that is going to be rectified/georeferenced. Also, you will notice that each major panel has three windows. These windows help you to zoom and navigate to the exact position you are interested in. The white square box also will help you to focus on the areas you are interested. You can click on the edges of this white squares to change the relative scene of the pane it represents. Take some time to orient yourself to zoom and pan the images.

- Left-hand side panel: Your Input image: Orbvirwsubset.img
- Right-hand side panel: Your reference image: Naipreferenceimage.img



12. Now you need to find some geographical landmarks in the NAIP reference image. So, zoom in and zoom out the reference image (right-hand side image) and select some landmarks such as road intersections, buildings, or other permeant structures. You should not select structures that may change their appearance with time, such as river banks, trees..etc.

13. Try to identify exactly the same landmarks in the left-hand side image as well. You can utilize options available in the **MGC window** to browse the landmarks easily. Also, you can always right-click on each image and fit the image into the window by selecting the **"Fit Image to Window"** option.

14. Now select **Create GCP** option  from the **MGC tool** window. Now go to the left-hand side image panel (input:orbviewsubset.img) and select the highest zoom pane and click on the object you need to select.

15. Now go to the NAIP reference image (naipreferenceimage.img) panel and find the exact same point you selected on the input image. Click the **Create GCP** icon and click on the matching object in the **Reference pane** with the highest zoom.

16. Now you will see that you have created the GCP in the reference image, and the points have been added to the attribute table at the bottom of the **MGC window**. Your input point data added to **X Input and Y Input** columns, and your reference point data added to **X ref. and Y ref. Columns.**

17. You can continue the above steps and can add a minimum of 20 control points. Once you have enough control points, click **Solve Geometric Model with Control Points** icon Σ.

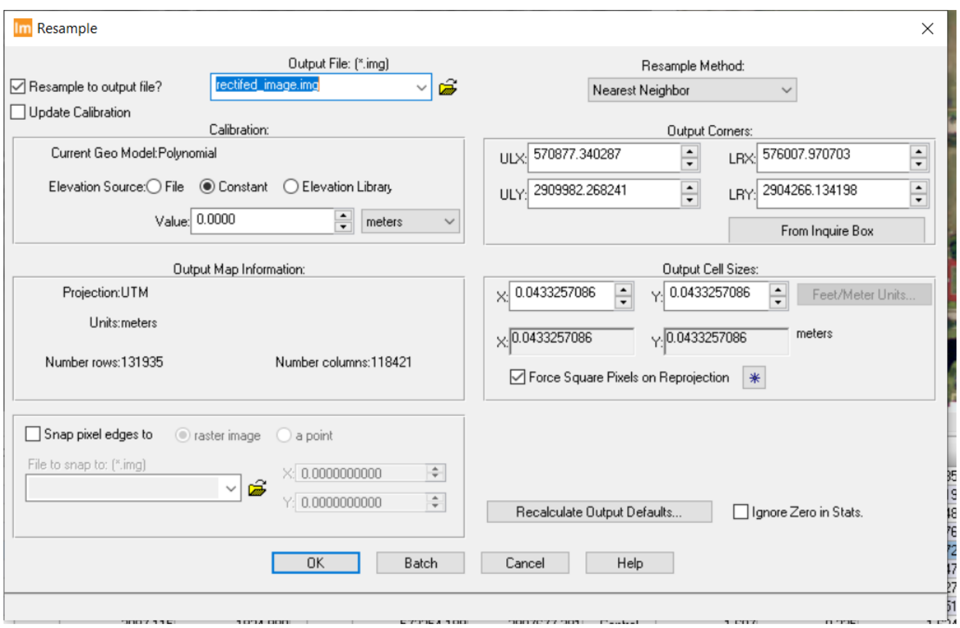
18. Also, you may need to save your work regularly. You can save your results by selecting options available in the **File menu**. Select **Save Input As…** to save points in the input image (left side of the screen) and select **Save Reference As**… to save the points on the reference image (right side of the screen). Save both files in the Lab 5 data folder with appropriate names.

Point #	Point ID	>	Color	X Input	Y Input	>	Color	X Ref.	Y Ref.	Type	X Residual	Y Residual	RMS Error	Contrib.
1	GCP #3			2706.737	-1522.759			572862.071	2908070.140	Control	0.530	-0.074	0.535	0.264
2	GCP #4			2083.810	-392.863			572216.326	2909183.594	Control	1.749	0.500	1.819	0.896
3	GCP #5			2916.981	-77.452			573066.130	2909520.277	Control	-0.821	-0.214	0.848	0.418
4	GCP #6			4570.081	-5198.792			574786.864	2904444.059	Control	-0.942	-0.253	0.976	0.481
5	GCP #7			3776.922	-4426.707			573973.639	2905198.381	Control	1.424	0.372	1.472	0.725
6	GCP #8			3106.762	-1949.390			573273.066	2907654.953	Control	-0.031	1.647	1.647	0.811
7	GCP #9			3140.035	-1949.990			573308.410	2907652.549	Control	0.704	-1.125	1.327	0.654
8	GCP #10			3590.786	-1818.610			573815.500	2907812.448	Control	0.723	0.206	0.751	0.370
9	GCP #11			3087.115	-1924.980			573254.189	2907677.391	Control	1.607	0.235	1.624	0.800
10	GCP #12			2723.595	-1729.227			572880.762	2907865.171	Control	-4.942	-1.294	5.109	2.517

NOTE: When you select the points, you may need to select the control points that cover the entire image. Also, check the **Root Mean Square Error** (RMS Error) column as well. If you can get a lower RMS error, it would be better for the results.
However, if you think you need to delete a point due to higher RMS value or if you think points are clustered and need to remove some extra points, you can select the respective row by clicking the **Point #** column. Then right-click and select Delete Section.

NOTE: X residual and Y residual give you an understanding of the contribution to the error by direction. So, you can decide which direction you should move your cursor to reduce the residual errors. **RMS value** is the resultant product of both X and Y residuals. Therefore, you may have to pick GCP's with lower X and Y residuals to get a lower RMS Error.

19. Once you have reached to the lowest RMS value that you can gain, now you can resample the input image pixels into geographic space. Click  icon to open the **Resample dialog** to specify the output file, resample method, and output information.
20. You can save the output file as **"resample_orbview.img"** by navigating to the Lab 5 data folder. Make sure to keep the resampling method is **Nearest Neighbor**, and you can leave other values as it is (default values). Click OK. Close the process list table once the rectifying task is completed.



NOTE: This step may take 5-10 minutes to complete (or more); once the progress is 100% complete, close the **Process List dialog** box and

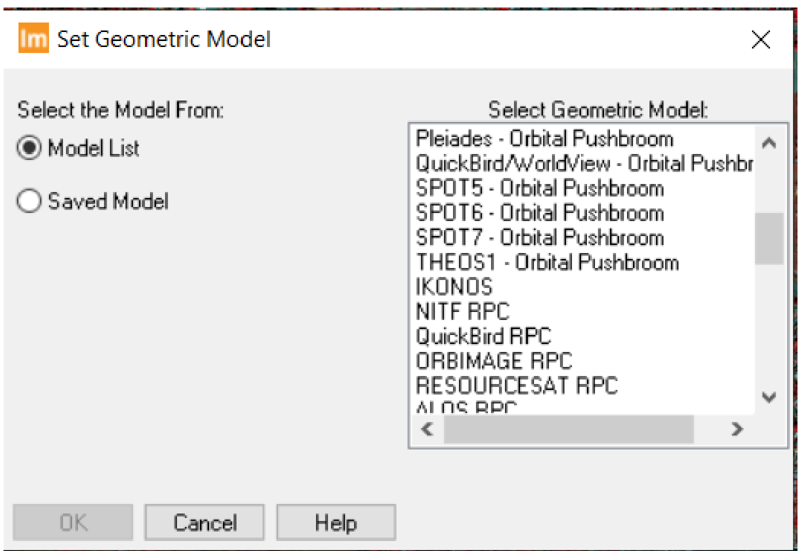
MGC window as well. You can save the output of MGC in the data folder with an appropriate name (e.g., **MGC_Orbviewmodel.gms**). Now you have successfully completed the rectifying step. You can load the rectified image to the main View by navigating to the Lab 5 Data folder and selecting **Open> Raster Layer.**

21. Add the **"resample_orbview.img"** to the main View and open the **Metadata** and check the projection information.

Q1: Include a screen-shot of the metadata dialog box together with "resample_orbview.img".

5.6.5 Orthorectification

1. Open a new window in **ERDAS IMAGINE** and add the image that you need to orthorectify. In this exercise, we will add Orbviewsubset.hdr image.
2. Navigate to **Raster| Panchromatic** and select the **Transform and Ortho group**. Click the drop-down menu and select **Ortho With Model Selection**.
3. Check **Model List** from Select the Model From. Then and select **GeoEye/OrbView - Orbital Pushbrrom** from the **Select Geomaertic Model** list. Click OK.



NOTE: GeoEye/OrbView - Orbital Pushbroom
Use this model for GeoEye or OrbView images in which RPC data is not available. This model is for data collected by GeoEye-1 satellite, developed by GeoEye, a company formed through the combination of ORBIMAGE and Space Imaging, as well as data collected by OrbView-3 satellite, built for Orbital Imaging Corporation (now GeoEye) (source: ERDAS IMAGINE (E.I.)).

Elevation Source for Orthorectification

1. Once you have completed the above steps, another dialog box will open that is related to **GeoEye/OrbView - Orbital Pushbroom RPC Model Properties.**

2. **Elevation Source** under the **General tab** allows you to set the elevation source for an image data file. This option is enabled for an image with a geometric model that uses an elevation value (Z).

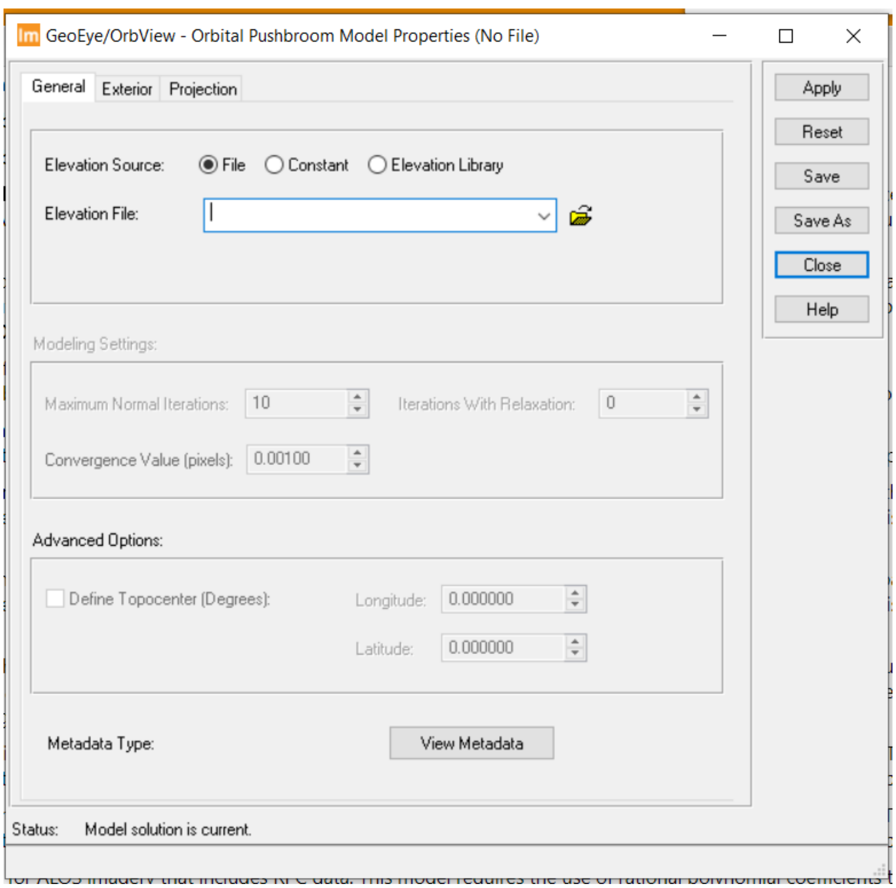
NOTE: This function is useful when you have several neighboring images because it will align them all to the same elevation source. You can set the elevation source to be a DEM file, a Z constant value, or use an Elevation Library.

NOTE:

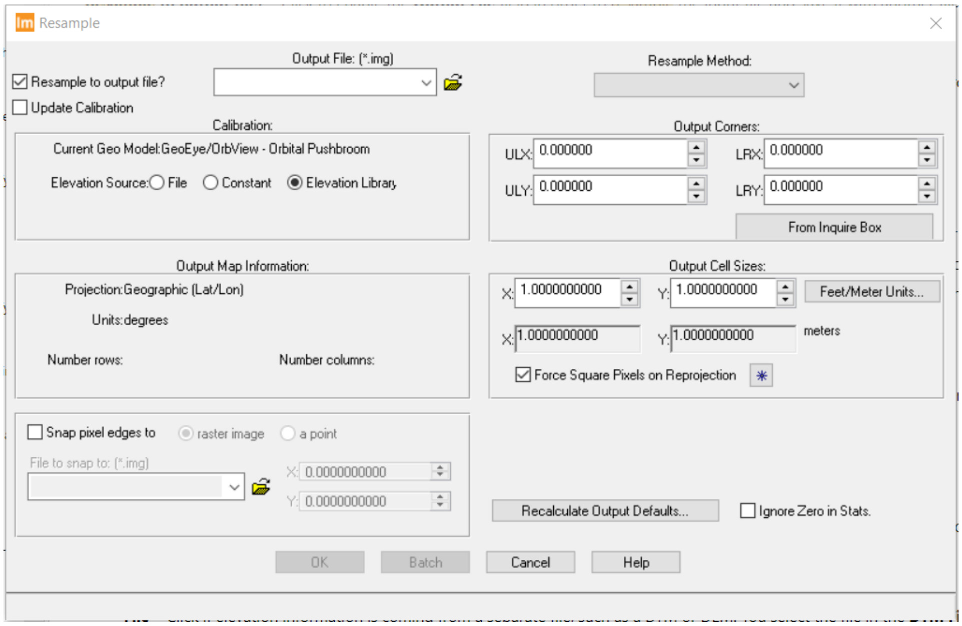
File → Click this radio button to obtain the elevation data from a DEM file. Select the File in the **Elevation File** field, below.

Constant → Click this radio button to use a constant value as the elevation information. You enter that value in the **Elevation value** field, below.

Elevation Library → Click this radio button to use the data in the Elevation Library as the source.



3. Now you can navigate to your **DEM file** (DEM.dat) location and add it as the elevation source. Click Apply
4. Next, Resample Window will open.



5. Check the Resample to output file option and name the File as“**resample_ortholab5.img**”

NOTE: If you want to attach a sensor model to an image without performing resampling, click to uncheck this checkbox. Then you can select **Update Calibration** option and write the geometric model information and elevation information to the *.aux file or image metadata as appropriate without producing a resampled image (E.I.).

6. Under the **Calibration** option, you can select **File** and navigate to the DEM file DEM.dat and add it. **Calibration solutions** report the current geometric model that you have assigned and allow you to select an elevation source for orthorectification.

NOTE: Elevation Source provides the elevation source to be used in the geo correction. By default, **Global Elevation Source proxy file**, stored in the Elevation Library, will be available.

7. You can select the **Nearest Neighbor** option as the **Resampling Method**. You can leave the rest of the features as default values and click OK.

NOTE: Nearest Neighbor → Assigns the value of the closest pixel.

Bilinear Interpolation → Uses the data file values of four pixels in a 2 x 2 window to calculate an output value with a bilinear function.

Cubic Convolution → Uses the data file values of 16 pixels in a 4 x 4 window to calculate an output value with a cubic function.

Bicubic Spline → Uses a block size of 5 x 5 or larger. This method fits a cubic spline surface through the current block of points. The output value is derived from the fitting surface that retains the values of the known points. This option may be available depending on which type of data is currently displayed or used in a process.

8. Now, navigate to the output file you just saved and open it in a new View. Compare it with the initial image. Add the **"resample_ortholab5.img"** to the main View and open the Metadata and check the projection information.

Q2: Include a screen-shot of the metadata dialog box together with "resample_ortholab5.img".

5.6.5.1 Orthorectifying Without GCP (form ERDAS IMAGINE HELP)

This workflow uses the method in which you do not have to load the image into a View. If the image is already loaded, you can still use this workflow.

Select the Geometric Model

1. Click **Raster** tab >  **Geometric Calibration > Orthorectify without GCP**.

The Geo Correction Input File Selector opens.

2. In File Selector, navigate to the image file to be orthorectified and select it. This File may typically be a TIFF file and has been rectified. Click **OK**.
3. The Set Geometric Model dialog opens.
4. If the specific sensor corresponding to your data is available, then click the **[specific sensor name] - Orbital Pushbroom** option. If the specific sensor is not available, click **Orbital Pushbroom**. Click **OK**.

Select the Elevation Source

The Orbital Pushbroom Model Properties dialog opens.

1. In **Elevation Source** option, click **File** to navigate to the elevation source file and select it.

Click **Apply**.

The Resample dialog opens.

Resample the Image

Resampling is the process of calculating the file values for the rectified image and creating the new File. All of the raster data layers in the source file are resampled. The output image has as many layers as the input image.

ERDAS IMAGINE provides these widely-known resampling algorithms: Nearest Neighbor, Bilinear Interpolation, Cubic Convolution, and Bicubic Spline.

See Resampling Methods section in Producer Field Guide for more information.

Resampling requires an input file and a transformation matrix by which to create the new pixel grid.

1. In Resample dialog under **Output File**, enter a name such as **[file_name_ortho.img]** for the new resampled data file. This is the output file created from orthorectifying the rectified File.
2. Under **Resample Method**, select **Bilinear Interpolation**.
3. In **Elevation Source**, note that the elevation source file name that you selected is shown in **DTM File** option.
4. Note the **Output Cell Size** values. The editable X and Y values are expressed in angular units of decimal degrees for a Geographic (Lat/Lon) projection. The non-editable X and Y values are the nominal output cell sizes expressed in meters or feet as indicated, according to the **Feet/Meters Units** option.

The software calculates recommended values, but you can change the values by clicking **Feet/Meters Units** button to open Nominal Cell Sizes dialog. In this dialog you can enter values expressed in meters or feet, and the software calculates the decimal degrees values.

5. Click **Ignore Zero in Stats.**, so that pixels with file values of zero are omitted when statistics are calculated for the output file.
6. Click **OK** in Resample dialog to start the resampling process.

The Process List dialog opens to let you know when the processes complete.

7. Close the Process List dialog when the job is 100% complete.

Verify the Orthorectification Process

One way to verify that the input image has been correctly orthorectified is to display the resampled image and the input image and then visually check that they conform to each other.

1. In a 2D View, load the input image.
2. Click **Home** tab > **Add Views > Create New 2D View**.
3. Open the orthorectified image in the second 2D View.

4. Click in 2D View #1 to make it active. Click **Home** tab > **Link All Views**. 2D View #2 is now linked to 2D View #1.

5. Click **Home** tab > **Inquire Cursor** to open an Inquire Cursor in both Views.

The Inquire Cursor (a cross-hair) is placed in both Views. An Inquire Cursor dialog also opens.

6. Drag the Inquire Cursor around to verify that it is in approximately the same place in both Views. Notice that, as the Inquire Cursor is moved, the data in the Inquire Cursor dialog are updated.
7. When you are finished, close the Inquire Cursor dialog.

RPC Model

In this workflow you are going to orthorectify an image by using an RPC model, which is typically used when rational polynomial coefficients (RPCs) are provided with the data. The RPC file contains a mathematical model of the image geometry, such as exterior orientation of the satellite when the data was collected.

The RPC file contains rational function polynomial coefficients that are generated by the data provider based on the position of the satellite at the time of image capture. This File should be located in the same directory as the image(s) you intend to use in orthorectification.

In the case of an NITF file, the **RPC File** field displays the image you are currently using in geometric correction because the NITF file contains RPC data internally. You can use an external RPC file if you wish.

There are various RPC models available that use specific sensor systems.

Select the Geometric Model

1. Click **Raster** tab > **Geometric Calibration > Orthorectify without GCP**.

The Geo Correction Input File Selector opens.

2. In File Selector, navigate to the image file to be orthorectified and select it. This File may typically be a NITF file. Click **OK**.
3. The Set Geometric Model dialog opens.
4. If the specific sensor corresponding to your data is available, then click **[specific sensor name] RPC** option. If the specific sensor is not available, click **NITF RPC**. Click **OK**.

The RPC Model Properties dialog opens.

5. Note the **RPC File** field.

This field displays the RPC file associated with the image. The RPC file contains rational function polynomial coefficients that are generated by the data provider based on the position of the satellite at the time of image capture. This File should be located in the same directory as the image(s) you intend to use in orthorectification.

In the case of an NITF file, the RPC File field displays the image you are currently using in geometric correction because the NITF file contains RPC data internally. You can use an external RPC file if you wish.

Select the Elevation Source

1. In **Elevation Source** option, click **File**. This opens an **Elevation File** field where you can navigate to the elevation source file and select it.

Click **Apply**.

The Resample dialog opens.

Resample the Image

1. Follow the steps in Resample the Image section.

Verify the Orthorectification Process

1. Follow the steps in Verify the Orthorectification Process section.

Orthorectify the Active Image

To orthorectify an image that is loaded in a View, you choose a workflow depending on whether or not your sensor model is associated with the image.

Sensor Model is Associated

If the sensor model is associated with the image (calibrated), follow this workflow:

1. Click **Multispectral**, **Panchromatic**, **Thematic**, or **Relief** tab (depending on the image type) > **Transform & Ortho** ▼ > **Ortho Using Existing Model**.

The Resample dialog opens.

2. Select the Elevation Source file in Resample dialog.
3. Follow the steps in Resample the Image section.
4. Follow the steps in Verify the Orthorectification Process section.

Sensor Model is Not Associated

If the sensor model is not associated with the image (not calibrated), follow this workflow:

1. Click **Multispectral**, **Panchromatic**, **Thematic**, or **Relief** tab (depending on the image type) > **Transform & Ortho** ▼ > **Ortho with Model Selection**.
2. The <u>Set Geometric Model</u> dialog opens.
3. Click the respective sensor model for your image, depending on whether the image has an RPC file associated with it or not. Click **OK**.

Specify your Elevation Source file in <u>RPC Model Properties</u> dialog (for RPC data), or in <u>Orbital Pushbroom Model Properties</u> dialog (for non-RPC data). Click **Apply**.

The <u>Resample</u> dialog opens.

4. Follow the steps in Resample the Image section.
5. Follow the steps in Verify the Orthorectification Process section.

5.6.6 Registering an image to a vector map

Vector data are stored as lists of coordinates, rather than as raster grids of numbers. These coordinates can be used to define points, lines, and polygons. We are interested in vector data because map data, such as roads and forest cover, are usually stored in vector format. Vector layers that are already projected to represent specific reference/coordinate systems can be used as a reference in the image to map registration.

To perform image to map registration process, ground control points (GCPs) are selected using the full resolution and Zoom windows. Coordinates are displayed for both base and uncorrected image GCPs, along with error terms for specific warping algorithms. Warping is performed using rotation, scaling, and translation (RST), polynomial functions (of order 1 through n), or Delaunay triangulation. Resampling methods supported include the nearest neighbor, bilinear, and cubic convolution. A comparison of the input and reference images using multiple Views allow quick assessment of registration accuracy.

The exercise below concentrates on just one form of Image to Map registration.

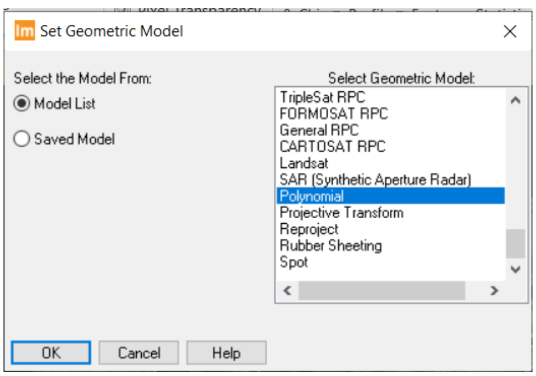
1. In the **File Tab**, click the Open button, and then **Raster Layer** option in **2DView #1.**
2. Navigate to the Lab 5 directory where you saved the Lab data. Select and open "**bldrspot.hdr**" file.
3. Now right click on the image and select **Fit to Frame** to see the entire image. Then click on the **Metadata** option in the **Home tab** so that you can view the image metadata.
4. Once the Image Metadata dialog box is open, check the Projection Info: You will notice this image is missing spatial reference information.

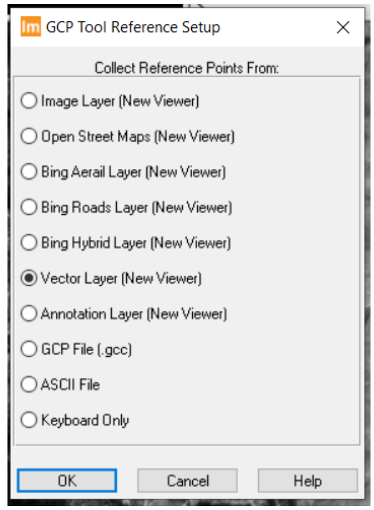
In the next steps, we are going to add spatial information to this image with the rectification process. Check if any other ERDAS windows opened. If so, close them.

5. Go to **Raster|Panchromatic Tab** and click **Control Points** option 

under **Transform and Orthocorrect** group.

6. Next, **Set Geometric Model** dialog box will open, select Model List, and scroll down the Select Geometric Model list. Select the **Polynomial model** from the list.





7. Next, **GCP Tool Reference Setup** dialog box will open and select the **Vector Layer** option and click **OK**.

NOTE: **Vector Layer (New Viewer)** is used to collect reference points from a vector layer loaded in the **Reference Views in the Multipoint Geometric Correction** workspace (source: E.I).

8. A new dialog box will open (**Reference Vector Layer**), and you need to navigate to your Lab 5 folder and select "**Roads. shp"** file and click OK. If you cannot see the Roads shapefile in the dialog box, you may need to select **"All File-based Vector Format"** from the drop-down menu under files of type.

9. Next, **Reference Map Information** dialog box will open. It will show you the Projection, Datum, and Horizontal Unit of the vector file. Click OK.
10. Then **Polynomial Model Properties** Box will appear. You can select the **Polynomial Order** as 1 by using the up-down arrows.

NOTE: If the **Apply** button is not activated yet, you can change the Polynomial Order to 2 and then the down arrow to return it to 1. This will allow you to be active in the Apply option.

11. Click **Apply** and close the dialog box. Now, you should get a separate window named **Multipoint Geometric Correction (MGC)**.

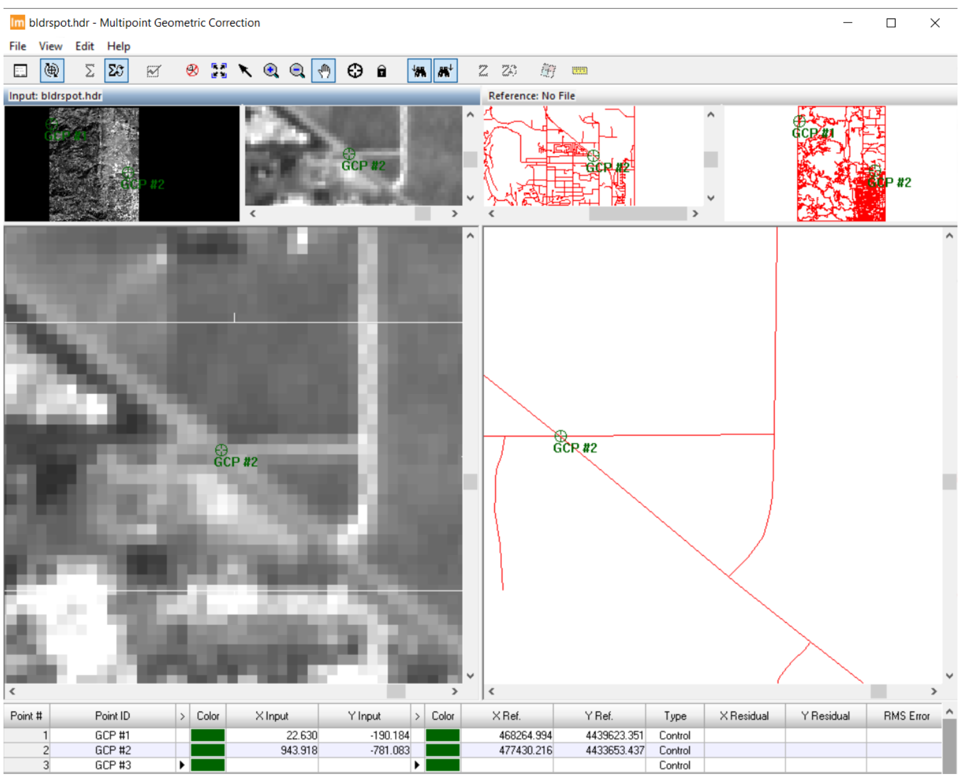
12. Now **Reference Vector file and Input image** have opened in the MGC window. You can see that the **Reference Vector File** is containing a road network. We can use this file to collect reference points for the georeferencing process.

13. Right-click on each pane (Left-hand side and Right-hand side) and select "**Fit Image to Window.**"

14. Before we start creating GCPs, we may have to change the color of the GCPs. Go to the **Color column** and select the first row. You will see an arrow mark for a drop-down menu to select a color. Select an appropriate color (do not add white). You can add color for GCPs in the reference image by following the same procedure as well.

15. You can click **Create GCP** ⊕ icon to create new GCPs. The cursor will turn into a cross-hair in the **View**. This tool must be locked to create GCPs repeatedly without having to click on this again.

    NOTE: 🔒 Click to unlock the tool currently in use. Once unlocked, you can enable different tools. 🔓 Click to lock the tool currently in use.



16. Next, you can select your first GCP in the input image by navigating to a road intersection that is clearly visible in both input and reference images. For this process, you can use zooming windows to select GCP accurately**.**

17. Once you have placed your GCP in the **Input image,** you can move to the reference window, find precisely the same road intersection, and place your GCP cursor on that intersection.

NOTE: You may have to check the nearby objects as well to confirm that you are selecting the same points in both images.

18. When you are selecting the GCPs, you need to distribute them evenly throughout the image. Continue the above steps and can add a minimum of 30 control points. Once you have enough control points, click **Solve Geometric Model with Control Points** icon Σ.

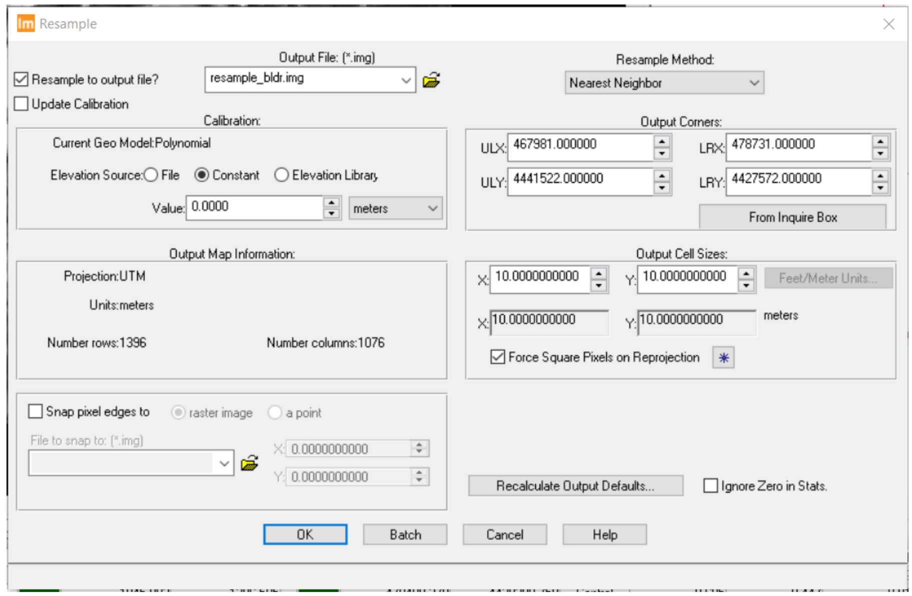
19. Also, you may need to save your work regularly. You can save your results by selecting options available in the **File menu**. Select **Save Input As…** to save points in the input image (left side of the screen) and select **Save Reference As**… to save the points on the reference image (right side of the screen). Save both files with an appropriate name in the Lab 5 data folder.

NOTE: When you select the points, you may need to select the control points covering the entire image. Also, check the **Root Mean Square Error** (RMS Error) column as well. If you can get a lower RMS error, it would be better for the results.
However, if you think you need to delete a point due to higher RMS value or think points are clustered and need to remove some extra points, you can select the respective row by clicking the **Point #** column. Then right-click and select Delete Section.
Also, You can move the GCP points over the image by clicking and dragging them to the place you need to move.

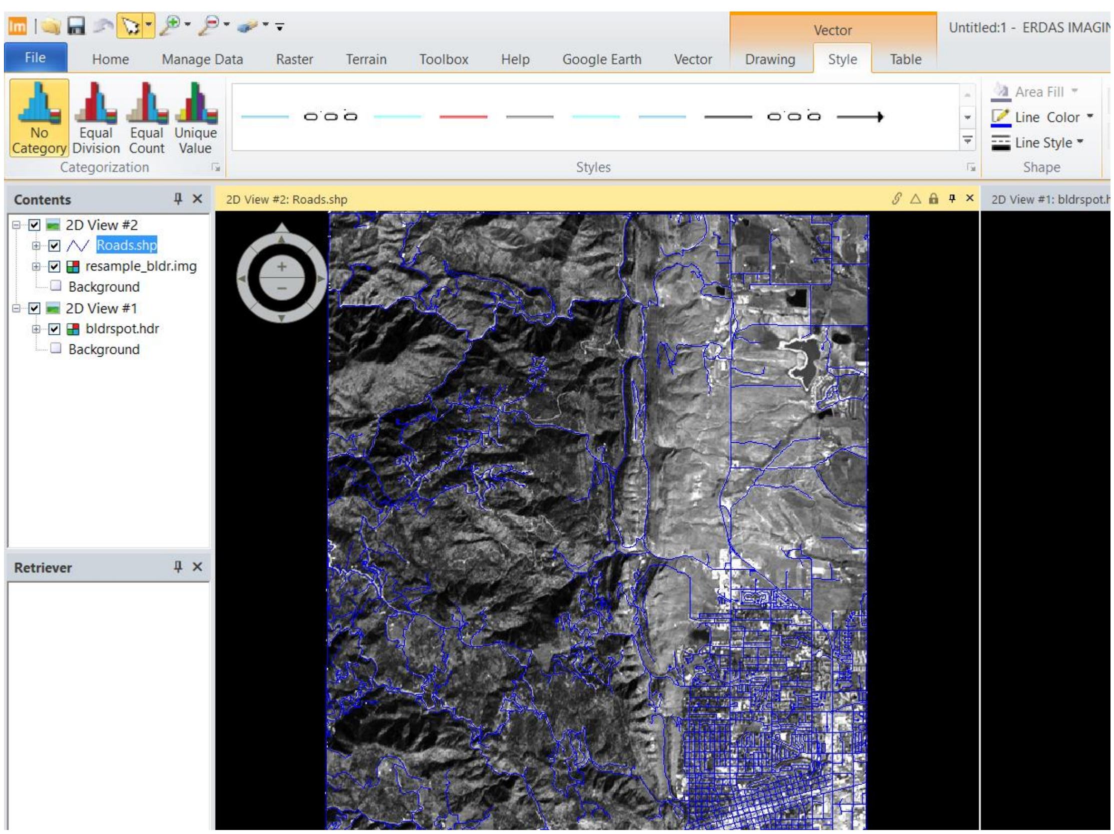
NOTE: X residual and Y residual give you an understanding of the contribution to the error by direction. So, you can decide which direction you should move your cursor to reduce the residual errors. **RMS value** is the resultant product of both X and Y residuals. Therefore, you may have to pick GCP's with lower X and Y residuals to get a lower RMS Error.

20. Once you have reached the appropriate number of GCPs, you can now resample the input image pixels into geographic space. Click  icon to open the **Resample dialog** to specify the output file, resample method, and output information.

21. You can save the output file as **“resample_bldr.img”** by navigating to the Lab 5 data folder. Make sure to keep the resampling method is **Nearest Neighbor**, and you can leave other values as it is (default values). Click OK. Close the process list table once the rectifying task is completed.



22. Now can go back to the ERDAS IMAGINE main window and add a new View. Once you added a new window, you can add “resample_bldr.img” that is just completed. Navigate to **Road.shp** file and add it to the same View.
23. You can see that the Road network overlaps with the actual road network on the “pldrspot.hdr” (Satellite image) image.



24. Click on **Road.shp** on the **Contents pane**. A new **Vector Tab** will be open. Go to **Vector|Style** and select appropriate color and line style so that you can see the road network clearly on the satellite image.

Q3: provide a snapshot of your rectified image with the road network. Also, provide a snapshot of the metadata window for the resample_bldr.img.

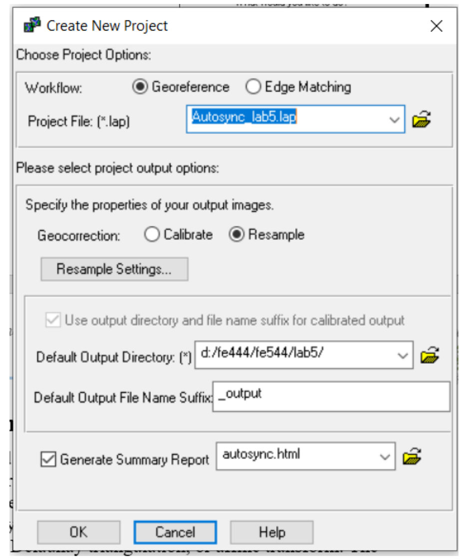
5.6.7 Image to image registration

Image to image registration, also know and image co-registration, is the process of geometrically aligning two or more images to integrate or fuse corresponding pixels representing the same objects. You can typically obtain the geometric relationship between an input image and a reference image through many tie points, and you can model the relationship using transformation capabilities, such as polynomial functions or affine transform. Run APM tool allows you to automatically generate tie points, thus making fully automatic image registration possible.

The IMAGINE AutoSync Workstation is a comprehensive tool to perform manual orthorectification and geo-correcting for images. Using this workstation, you can load input and reference images (when geo-correcting images), select a sensor model, run automatic point measurement (APM), rectify images, create an AutoSync summary report, and output image verification. You can also manage your AutoSync project, modify parameter files, and perform data visualization. Use the IMAGINE AutoSync Workstation for complex workflows since it provides more flexibility and tools for visual results inspection and manual digitizing. All areas of the workstation are resizable except for the menu bar and the toolbars (source: ERDAS IMAGINE).

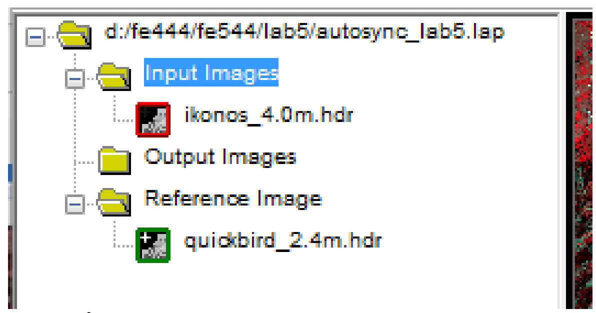
In this lab, you will use the IMAGINE AutoSync Workstation in different scenarios to geometrically align two overlapping images with different viewing geometry and different terrain distortions into the same coordinate system so that corresponding pixels represent the same objects. In the following steps, you will perform image registration by selecting input and reference images, setting parameters to generate tie points, reviewing and editing those tie points, and generating an aligned output image.

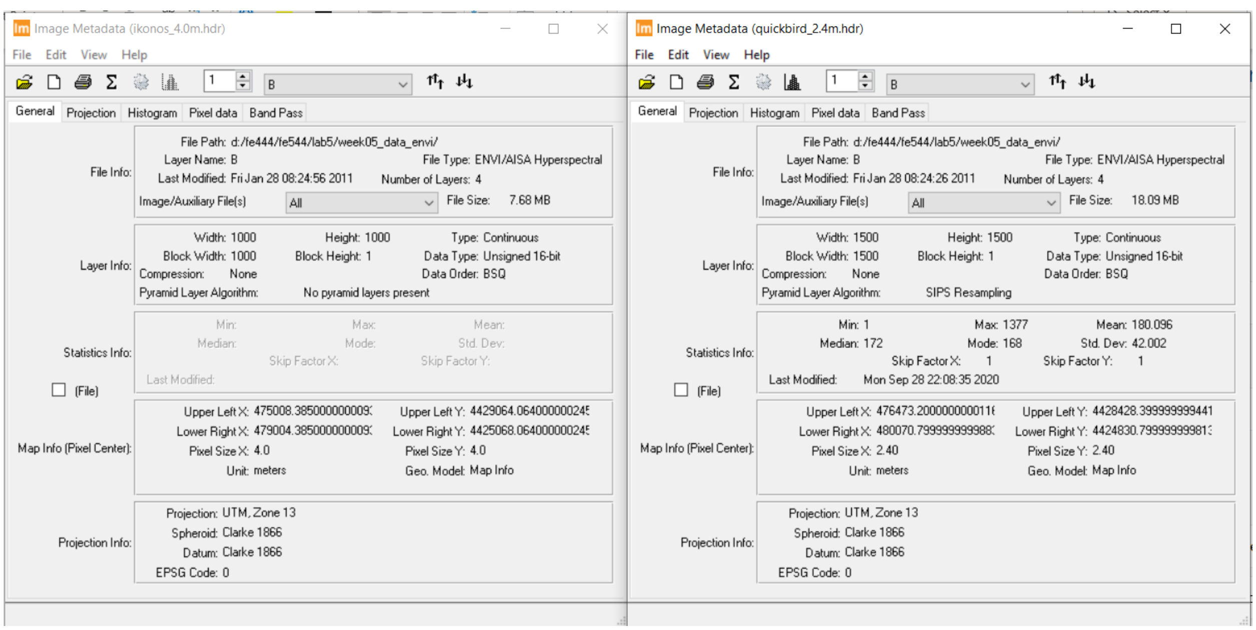
1. You can start the **IMAGINE AutoSync** process by navigating to **Toolbox Tab**. Now, select the **AutoSync Workstation** icon  and select **AutoSync Workstation** option.
2. A new dialog box will open. If you are doing this process for the first time, you can select **Create a new project.**
3. Now you will see the **Create New Project dialog** box, and you can select Workflow as **Georeference** and type **Project File** as **"Autosync_lab5"**. Then select the **Resample** option for **Geocorrection** and leave other settings as default.
4. You can go to **IMAGINE AutoSync Workstation** window and add the input image and the reference image to the project using the left-hand side panel.



Input image: ikonos_4.0m.hdr
Reference image: quickbired_2.4m.hdr

5. Now you can select the input image by right-clicking the **Input Images tab** and select Add **Input Image.** Navigate to your Lab folder and select **"Ikonos_4.0m.hdr" as the input image.**
6. Then you can select the **Reference Image** by right-clicking the **Reference Images Tab** and select **Add Input Image.** Navigate to your Lab folder and select **"quickbird_2.4m.hdr" as the reference image.**





NOTE: You can check the Metadata for both the images by selecting the icon in the Workstation window.

Q4: include the snapshot of Metadata for both the images.

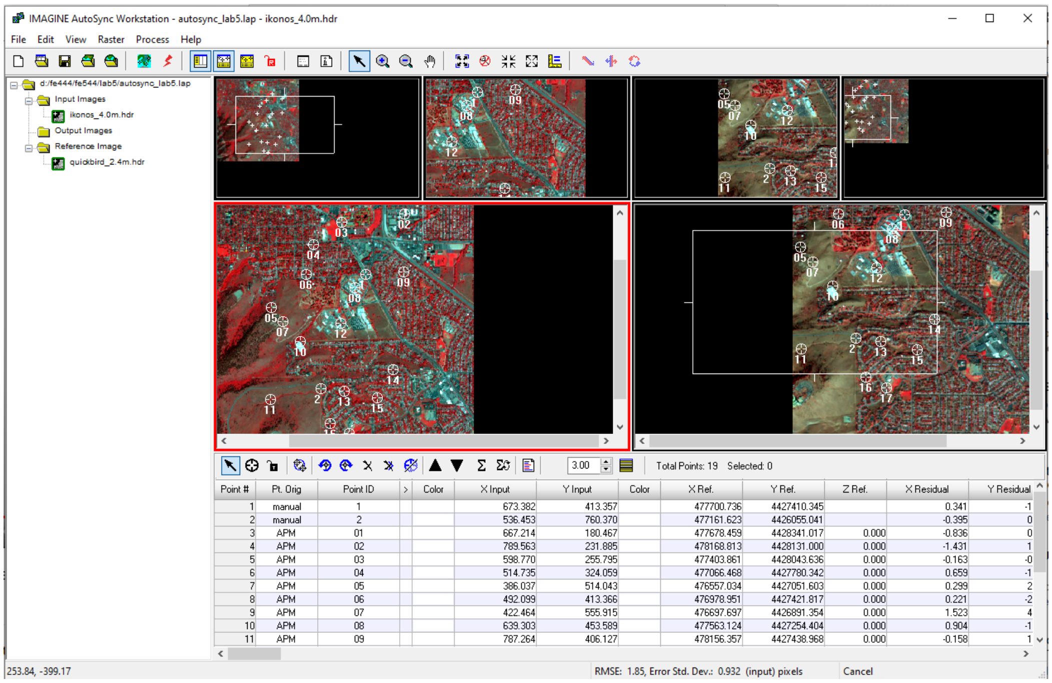
Now we will create tie points to establish the initial spatial correlation between the input image and the reference image.

7. You can choose specific tie points manually by selecting specific locations such as road intersections or corners of buildings.
8. First, select the **Create CGP** icon and select a road intersection in the input image and then select the same location in the reference image. You can select the magnifier windows to zoom locations you are interested in.
9. Add only two GCPs to be familiarized with selecting the points.

NOTE: When using the Georeferencing workflow, you can adjust the algorithm settings that control the placement of automatically generated tie points in your images on the APM Strategy tab. Automatic point measurement (APM) identifies similar features between the input image and the reference image. Depending on the quality and type of imagery, you may need to try different strategies to find a large number of high-quality points (source: ERDAS IMAGINE (E.I.)).

10. Now we can run the **APM by Selecting Run APM** option by **Clicking Process** from the **Imagine AutoSync Workstation menu bar.**

11. Now you will see automatically generated APM points in both input and reference images. Check the RMSE value and record the value.



12. Let us look at some important settings behind the **APM process**. Go to process and select Project Properties. **IMAGINE AutoSync Project Properties** dialog box will open.

13. Go to the **APM strategy** tab and check the **Starting Column, Starting Line, Column Increment, Line Increment,** and **Output Geometric Model Type under the Geometric Model**. Record this information. You need this information to answer the questions in the later part of this section.

14. **Now let's look at some features available in the APM strategy tab.**

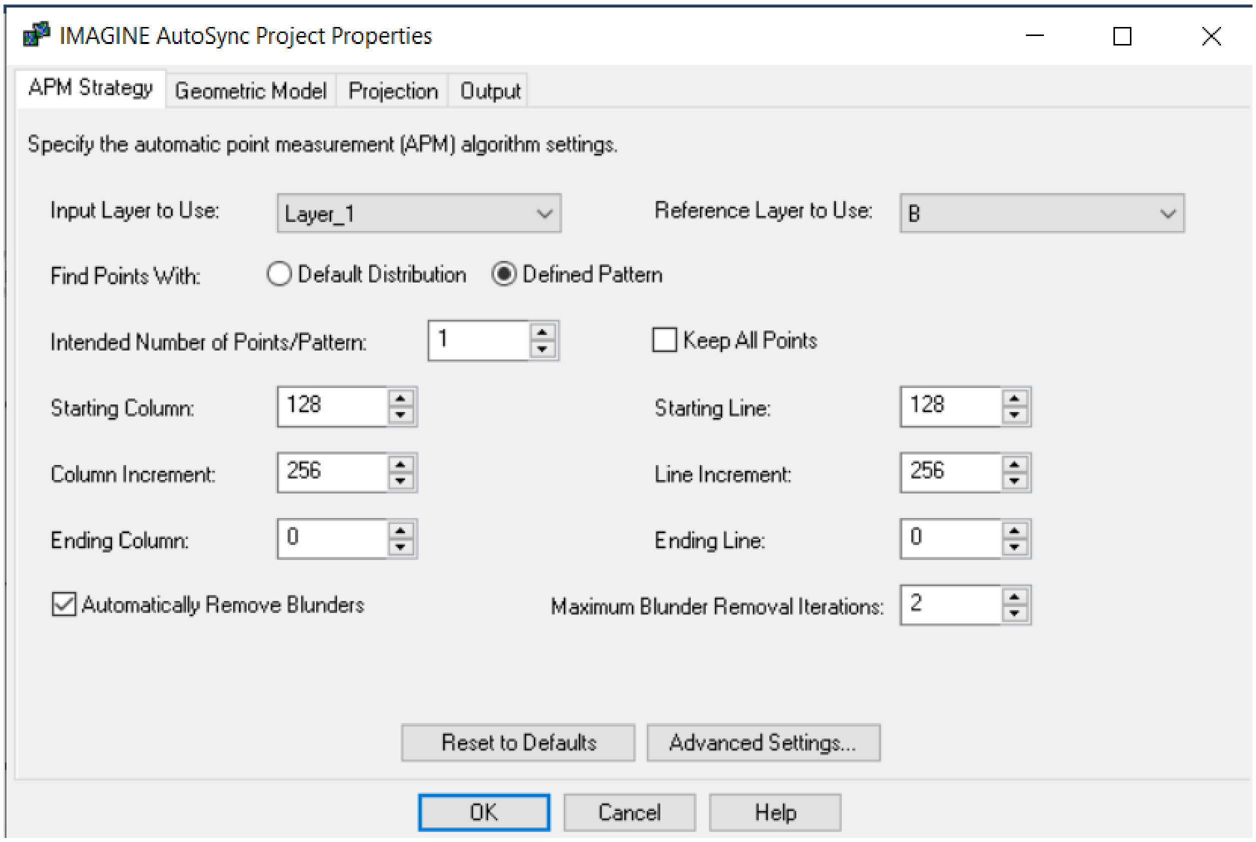
    Defined Pattern: This has many options, including **Starting Column**, **Starting Line**, **Column Increment**, **Line Increment**, **Ending Column**, and **Ending Line**. Using those options, you can define the exact placement of tie points throughout the images of the block. With this option, you also define the intended number of points per pattern.

NOTE: If you want to define a tie point pattern, select the **Defined Pattern**. However, you should consider the overlap percentage when you define your own pattern and try to get each pattern location inside as much of the overlap as possible (source:E.I).

Under Defined Pattern:

- **Starting Column and Starting Line have** setup the default value as 128. These values are important if APM does not find enough points along the edges of the overlapping area, then reduce the size of the **Starting Column** and the **Starting Line**.
- **Column Increment and Line Increment have set up the default value as 256. These values are important i**f the overlapping area between the images is not big

enough, or if APM does not find enough tie points, then reduce the size of the **Column Increment** and **Line Increment**.



15. Run the **APM process** with default values (click OK). Look at the RMS value and record it. Also, take a snapshot of IMAGINE AutoSync Workstation and save it to refer to the distribution of tie points later.

NOTE: You need to select (if it is not selected by default) **Automatically Remove Blunders** checkbox to remove blunders (wrong tie points) automatically from the APM generated tie points.
When this option is selected, the points that do not fit well with the majority of tie points are considered as blunders and are discarded. By default, this option is selected. You should deselect this option only if you suspect that it is removing correct tie points. For example, you should deselect this option when most of the APM tie points are wrong or when there is a large difference in the terrain between the two images.

Maximum Blunder Removal Iterations: This option becomes available when you choose **Automatically Remove Blunders** option. The default is 2. In most cases, increasing this number means more iterations of the blunder removal algorithm will be run. As a result, more tie points will be considered as blunders and discarded (source: E.I.).

16. Now, you can do the APM process again by changing the default values. Try to assign **the Starting Column and Starting Line values to 64 and Column Increment and Line Increment values to 128. Check the RMS value and distribution of tie points. Record the RMS value and save the snapshot of IMAGINE** AutoSync Workstation.

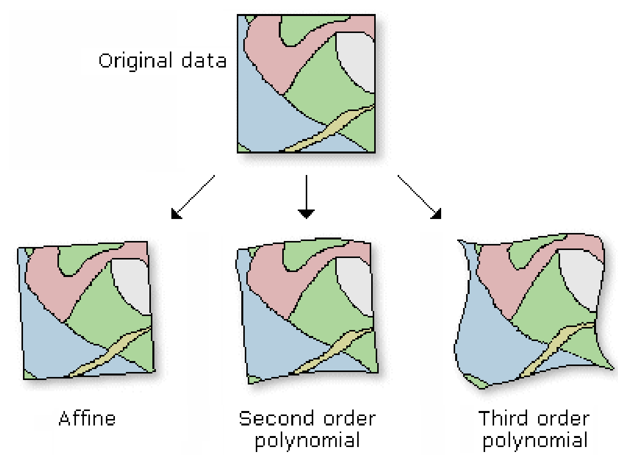
Q5: Describe the observations you just made by changing the values, particularly how the RMS values changed. You can add saved snapshots to illustrate your answer.

17. Now navigate to the Geometric Model tab on the **IMAGINE AutoSync Project Properties**. You can do this by **Clicking Process > Project Properties...** from the IMAGINE AutoSync Workstation. Then, click the Edit Project Properties icon on the toolbar.

18. You well see the **Output Geometric Model Type** as **Affine** transformation by default. This Affine geometric model allows the image to be flipped, rotated, or scaled. The results you obtained so far are related to Affine transformation.
19. Now, you can try to change the Output Geometric Model by selecting the option available in the drop-down menu. Select the **Polynomial option**. By selecting this option, you can use the polynomial function to perform surface approximation.

NOTE: The polynomial geometric model uses polynomial coefficients to map between image spaces. This option corrects the remaining error and refines the mathematical solution. This is the fastest algorithm but does not produce the best results.

20. Once you have selected the **Polynomial geometric model** type, you will see the **Maximum Polynomial Order** option.



NOTE: This is the maximum polynomial order allowed for polynomial approximation. The 1st order is an affine transformation. The 2nd order results in a second-order transformation; the 3rd order a third-order transformation (source: E.I).

The higher the transformation order, the more complex the distortion that can be corrected. However, transformations higher than third order are rarely needed. Higher-order transformations require more links and, thus, will involve progressively more processing time.

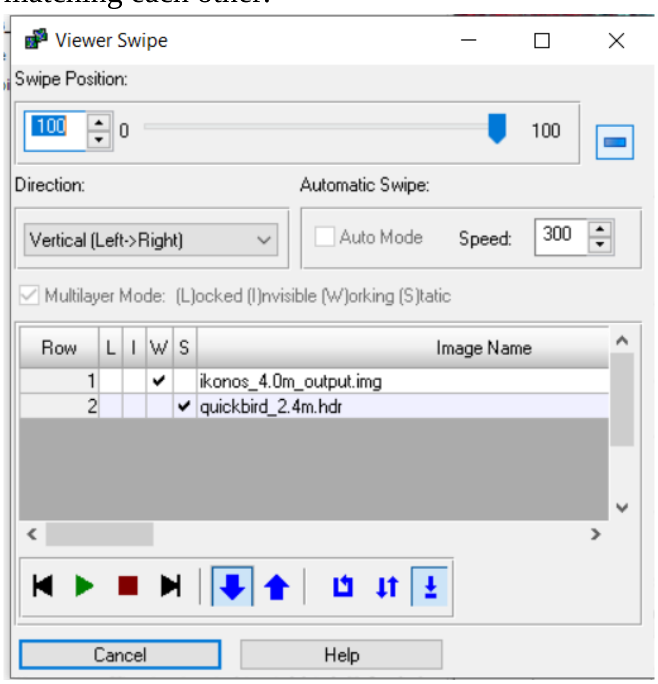
In general, if your raster dataset needs to be stretched, scaled, and rotated, use a first-order transformation. If, however, the raster dataset must be bent or curved, use a second- or third-order transformation.
(Source: ESRI ArcMap, Fundamentals of georeferencing a raster dataset: https://desktop.arcgis.com/en/arcmap/10.3/manage-data/raster-and-images/fundamentals-for-georeferencing-a-raster-dataset.htm).

21. If you need to use a higher order of transformations, you need to have more GCPs. In order to achieve the best rectification results, you may need to collect more than the minimum number of GCPs, and the GCPs need to be well-distributed and as precise as possible.
22. If you do not observe improvement after you run the polynomial model by selecting the Σ icon or **process> Solve models,** you can change the **Maximum Polynomial Order** into 2. Report your RMSE value. Have you noticed an improvement in RMSE value?

23. Alternatively, you can sort the Error column and can delete the rows with higher error. Once you delete those tie points, you can re-run the model and observe the RMSE value.

24. To finish this process, you can right-click on **"ikonos_4.0m.hdr"** on the left-hand side panel, and select **Calibrate/Resample.**
25. Now you see the output image and reference image have been added to the main window. Also, the output image is added to the lefthand side pane under the **Output Image** option.

26. So, we can do some qualitative assessment by comparing the output image and the input image by utilizing the **Start Swipe Tool** in the toolbar.
27. You can move the **Swipe Position** bar to see how the input image and output image matching each other.



Q6. Include a snapshot of your Output image with your reference image.

6 Image Classification

Lab developed by Rong Fang and Dr. Strimbu from the notes of Dr. Hilker and ENVI documentation.

***Note**: throughout this lab, the terms class, category, and land-cover type can be taken to mean the same thing and are used interchangeably.*

6.1 Objectives

- Define areas of interest (ROI)
- Exploring spectral properties with 2D Scatter Plots
- Execute supervised classification of images
- Assess the quality of the classification

6.2 Files to be used in the Lab

- LandsatTM is a Landsat 5 image of the Gulf Islands from south BC acquired in 2006.

Before beginning the lab, you should visit the last page and examine the Submission section.

6.3 Supervised Classification

Supervised classification of remotely sensed data begins with delineating boundaries around groups of spectrally similar pixels, known as “training areas”, which are representative of specific land-cover types. The location of training areas might be determined in the field using GPS; interpreted directly from other forms of remotely sensed data (e.g., aerial photography); or from other sources available for use in GIS software (i.e., existing digital maps). Training areas (i.e., groups of spectrally similar contiguous pixels) representing specific land-cover classes, once delineated, are used to calculate statistics characterizing the distribution of pixels’ Digital Numbers (DN) in all bands/channels representing each particular land-cover type. Some of these statistics include Minimum, Maximum, Mean, and Std. Dev. These statistics are used to assign membership to a class/category (land-cover type) for each and every image pixel through the application of an image classification algorithm (e.g., Maximum Likelihood).

In summary: groups of spectrally similar contiguous pixels are delineated and act as training areas representing specific land-cover types/classes/categories. The digital numbers/values associated with these training areas are then summarized. The summaries facilitate assigning membership to all other pixels in an image. Membership for each individual pixel is assigned based on the values representing a particular class that are closest to the values contained in an unclassified pixel. The end result is a classified image displayed with colors assigned to represent each class/category/land-cover type.

6.3.1 Image Overview

The data used in this lab is a Landsat TM image acquired on July 23 2006. The image contains six bands: b1 (blue); b2 (green); b3 (red); b4 (near-infrared (NIR)); b5 (shortwave infrared (SWIR)) and b7 (SWIR).
Q1 [10%]. Landsat band 6 is not included in the layer stack and is therefore purposefully excluded from this classification exercise. What portion of the electromagnetic spectrum does this band represent, and why do you think it is not included (two sentences)?

From the available bands list, display an RGB color composite, using bands 1 (blue), 2 (green) and 3 (red).

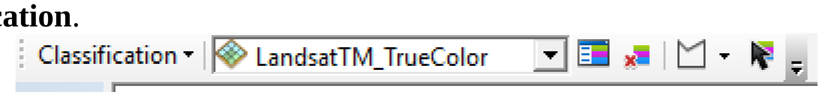
Q2 [10%]. What is a true color composite (one sentence)? What region of the electromagnetic spectrum defines it (one sentence)?

Open ArcGIS, and save your project immediately, even that is empty. Add the *LandsatTM* image from Week6_Data. Set the display band as ***Red: band 3, Green: band 2, and Blue: band 1***. Change the layer name to "LandsatTM_TrueColor", by right-clicking on the layer, and choose **Properties → General**, then typing *LandsatTM_TrueColor* in the Layer Name box.

Add Base Image provided by GIS as reference. Click → **Add Basemap → Imagery → Add**. The **Basemap** layer will appear in the **Table of Contents**. Right-click on the LandsatTM_TrueColor layer, choose **Zoom to Layer**. The Basemap contains layers with a resolution of 15m, 60cm, and 30cm.

6.3.2 Collecting training area of classes

1. To classify an image using supervised methods, first, you must start the Classification menu. Therefore, in the main menu, choose **Customize → Toolbars → Image Classification.** Make sure **LandsatTM_TrueColor** appears in the drop bar of **Image Classification**.



Supervised classification requires identification of training areas. We are using the **Base Image** provided by ArcGIS as a reference in the delineation of training and testing areas. DO NOT define the training areas in the Base Image itself, because this may take up too much memory. The polygons should be drawn in the Landsat imagery. If the *LandsatTM* layer is under the Base Image, drag it above the **Base Image** in **Table of Contents**.

2. To best utilize the Base Image as reference, open **Effect toolbar**, in the main menu **Customize → Toolbars → Effects,** set *LandsatTM_TrueColor* as the active image in the Effect toolbar. Use the **Swipe tool** to swipe the Landsat image to compare it with the Base Image.

To classify the Landsat TM image, we will delineate polygons for several land-cover categories. There is no specific rule dictating how many land-cover categories should be created; however, bear in mind that a land-cover class is only realistic if it is spectrally unique and observable at the spatial resolution of the imagery. In other words, the spatial and spectral resolution of Landsat imagery in most cases allows for distinguishing between taxonomic tree types, such as broadleaf and coniferous, but do not permit separating specific tree species, such as Douglas-fir and Western redcedar. Therefore, defining broadleaf and conifer as target land-cover categories is practical and achievable, as these are realistic spectral classes, whereas classes based on specific tree species (e.g., Douglas-fir and Western redcedar) are unrealistic as these information classes (i.e., species type) are not spectrally distinguishable by the Landsat TM sensor.

When considering Landsat imagery, beyond forests, other types of vegetation can be distinguished. For instance, oftentimes, areas dominated by herbs, shrubs and/or agricultural growth will appear spectrally different to forests in Landsat imagery; however, these non-forest vegetation types may appear spectrally similar to each other. An exception to this relates to vegetation health, which can be associated with things such as stress (due to water shortage and/or pest infestation). Furthermore, the time of year is important. Due to phenological properties, vegetation may appear very differently in July and January.

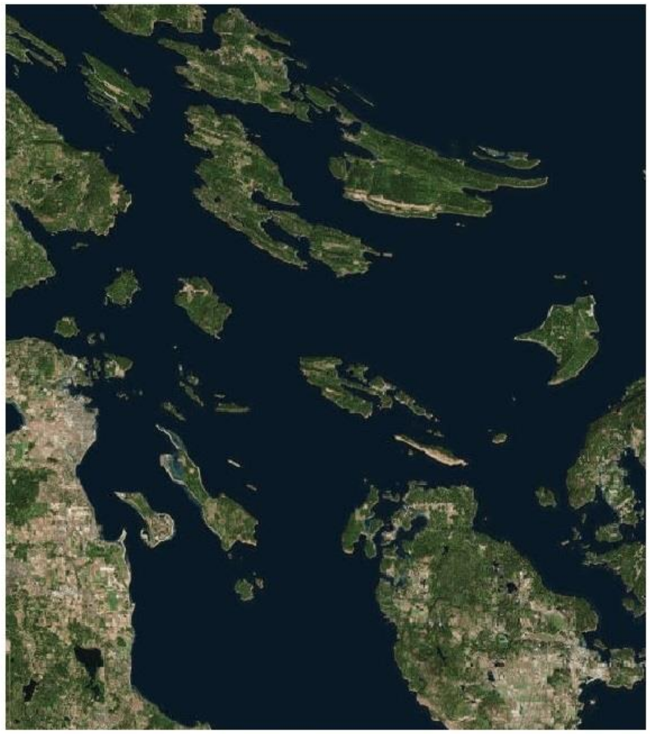
Beyond vegetation, other types of land-cover features are spectrally identifiable in Landsat imagery. These may include classes relatable to water and/or man-made features.

Q3 [10%]. When classifying images using spectral information, what characteristics of the image are not considered explicitly (one sentence)? To answer this question, think back to the advantages provided by remote sensing, such as nadir view or information on wavelength invisible to the human eye.

3. To distinguish vegetation from the rest of the land cover better, add *LandsatTM* again, and display it as false color. You should set ***band 4 as red, band 3 as green, and band 2 as blue.*** Name the newly added false-color image as ***LandsatTM_FalseColor***. Compare the different land covers (vegetation, water, urban, pasture, etc.) in the three images (Landsat true color, Landsat false color, and base image).

RGB: 321 True Color Landsat TM image

RGB: 432 False Color Landsat TM image

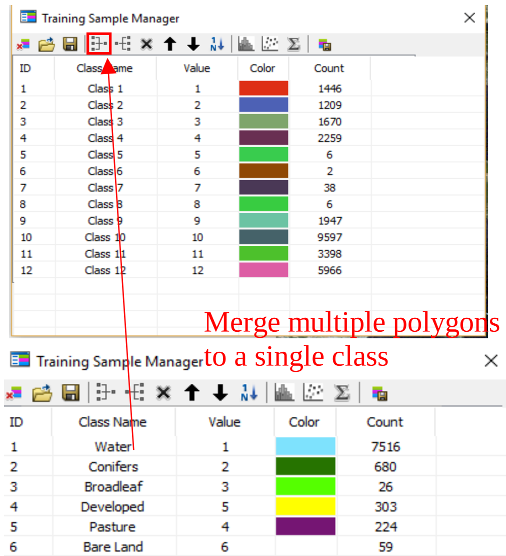


ArcGIS Base Image

4. To begin the classification process, in the **Image Classification** toolbar, click on **Draw Polygon** , which will turn on the drawing option. If the true-color image is not used than make sure that is selected. If the polygon option is not selected, then choose to **draw polygon** from the drop-down menu. Navigate to an area of interest (that you belief to represent a specific land-cover category). Also bear in mind that your **Draw Polygon** only corresponds to one of your active images displays.

5. Once you have outlined the shape/perimeter of a feature/area of interest, double clicks will close the polygon. If you are unhappy with the polygon that you have just completed, at any point, you can delete the polygon by clicking **Clear Training Samples icon** in the image classification tool. Keep delineating more training areas for one class. To see all the polygons belong to different classes, you should click the **Training Sample Manager** icon in the Image Classification tool, which will open the Training Sample Manage. To create a class from the delineated polygons, you must group them.

6. Select all the polygons belonging to one class by simultaneously pressing Shift and the left mouse button. Once all the polygons were selected, press **Merging Training Samples icon** , and all the classes will be integrated into one. It is good practice to change the **Class Name**, **Value,** and **Color** of the class that you just created to something that is easy for you to identify. Repeat these steps to create at least six classes. For example, the six classes could be **Water**, **Conifers**, **Broadleaf**, **Grass**, **Developed** area, and **Bare land**. If they were delineated in that order, the corresponding indexing values would be 1, 2, 3, 4, 5, and 6.



Q4 [10%]. What properties of vegetation are mainly responsible for reflective/absorptive characteristics in the red and the NIR (one sentence)?

NOTE: SAVE YOUR FILE OFTEN! In Training Sample Manager, click then, save the training polygons as a shapefile. If you close the program, the polygon file with the selected training area will still exist.

You can also use the help from World Imagery to identify the land use/land cover class and delineated the polygons if it is difficult to identify the objects.

• For every land-cover class, you need multiple polygons. It is better (and required!) to have numerous small class-specific training areas spread out throughout the image than it is to have only a couple of very large training areas.

• To capture class variability, you should have at least 10 different polygons per class as evenly spread throughout the image as possible.

Q5 [10%]. Why it is recommended that the training areas to be spread across the image?

• The total number of pixels occupied by all polygons in a class should be at least 100, but in general, it should be much larger (i.e., > 1000). Remember that the number of pixels is class-specific.

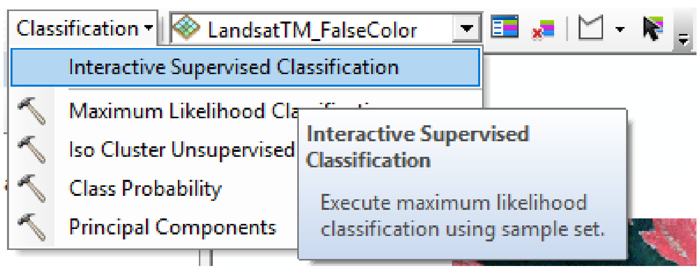
• For vegetation classes and water, you are REQUIRED to have 10 polygons per class totaling > 250 pixels (per class).

• For all other classes (i.e., barren and exposed; low density developed; high density developed; and any additional classes you may have defined) you are required to have 10 polygons per class with at least 100 pixels per class.

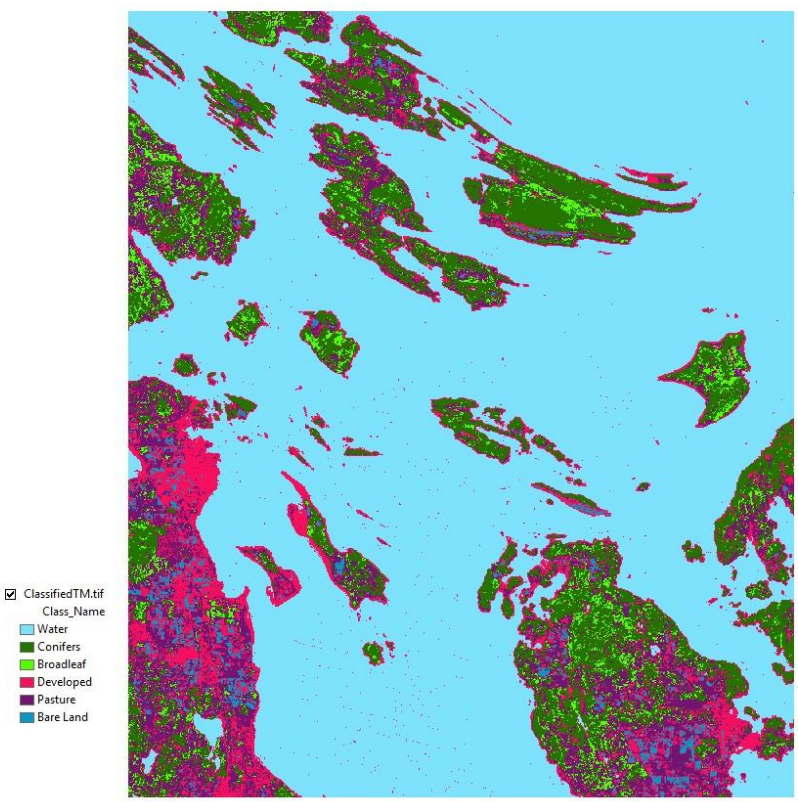
Q6 [10%] How many land-cover categories are you using? What are the areas described by each (i.e., water, trees, grass, buildings, bare, etc.)? Provide a screenshot of your training polygons distribution in your final submission.

6.3.3 Image Classification

Once the training areas are delineated, you can classify the Landsat TM image. In this lab,you will be using the **Maximum Likelihood** classification algorithm and all 6 bands as input. Save the final selected training polygons as the **LandsatTM_class.gsg file**. By clicking on the Create Signature file icon available in the Training Sample Manager window. In the drop-down list of the Image Classification tool, choose **Interactive Supervised Classification.** Classification will be executed with maximum likelihood by using the sample data. The classified layer will be automatically added in the Table of Contents.



Q7 [10%]. Provide a screenshot of your classified image of the true color image, with Table of Contents showing the classes as below:

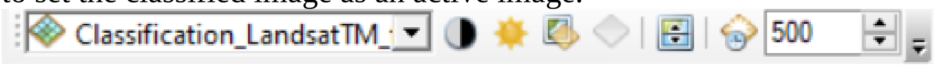


Q8 [10%]. Briefly describe (2 sentences) the nature of Maximum Likelihood classification.

6.4 *Classification Accuracy*

6.4.1 Visual examination

We will now explore the quality of the classified image. For visual purposes, you can change the colors assigned to your classes to make the classes more distinguishable. Use the **Effects toolbar** to set the classified image as an active image.



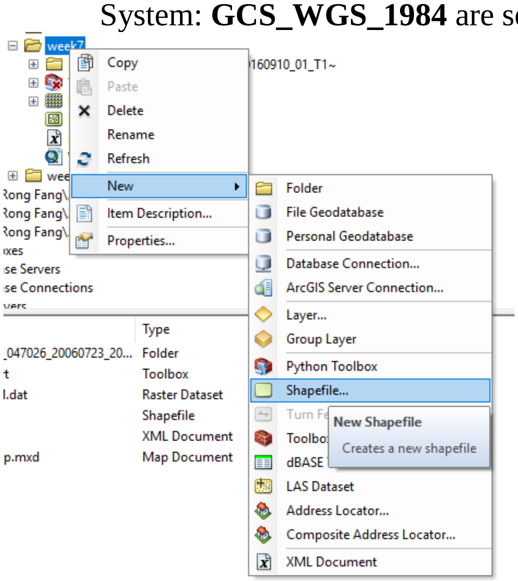
1. Now, navigate around the classified image and compare it to the original image and the false-color image. Ask yourself: was this a successful classification? Does the distribution of classes make sense? Are there any areas of major misclassification? Would you feel comfortable and confident passing along this map to someone who would then make "real world" decisions based on it?

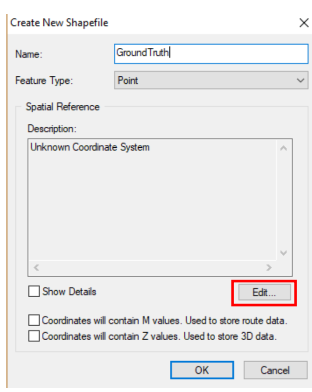
Q9 [10%]. Justify the visual quality of the image by describing two issues and two successes of the classification (four sentences).

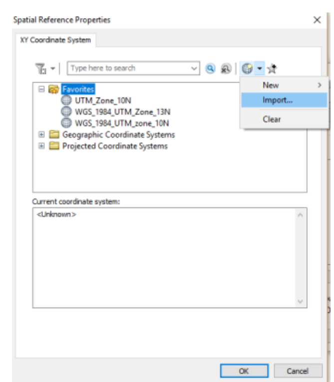
6.4.2 Formal assessment of accuracy using Ground Truth Regions of Interest

To formally assess the accuracy of the classification with the ground truth sample, we need to create a layer of ground truth, which will be compared with the classified image.

2. Open ArcCatalog , and navigate to the week 6 folder. Right-click the folder, choose, and **New → Shapefile**. Name the shapefile as ***GroundTruth*** and select **type point**. Set the coordinate system the same as the Landsat image. Click **Edit**, and click to choose **Import**. Navigate to folder were you save the Landsat TM image and select LandsatTM. Click OK. Projected coordinate system: **UTM_Zone_10N** and Geographic Coordinate System: **GCS_WGS_1984** are selected. Click **OK**.

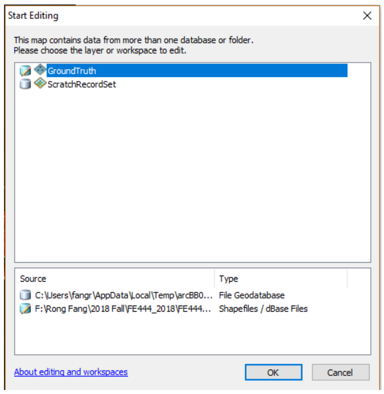


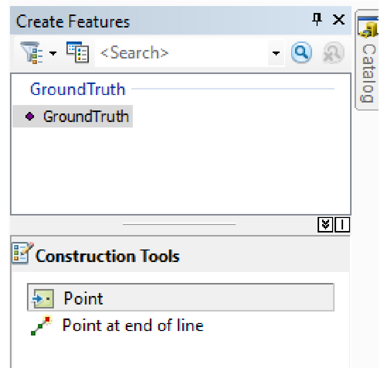




Next, we need to point the ground truth sample data. Let's take the class Water as an example.

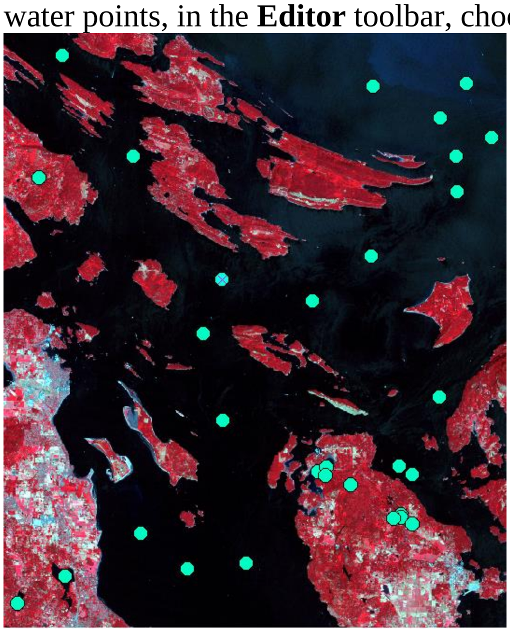
3. **Enable the edit of GroundTruth layer**.
   Open Editor Toolbar: in the main menu, choose **Customize → Toolbars → Editor**. In the drop-down list of Editor, select **Start Editing**. Choose **GroudTruth** as the layer for editing. And click **OK**. In the editor toolbar, click **Create Features icon** . In the **Create Features** window, click **GroundTruth**, and in the **Construction Tools**, choose **Point**. Now you are able to choose the reference points on the top of the **LandsatTM** image.

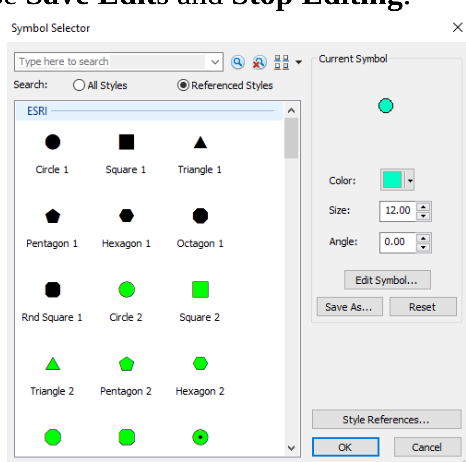




4. Choose reference ground truth points of water.

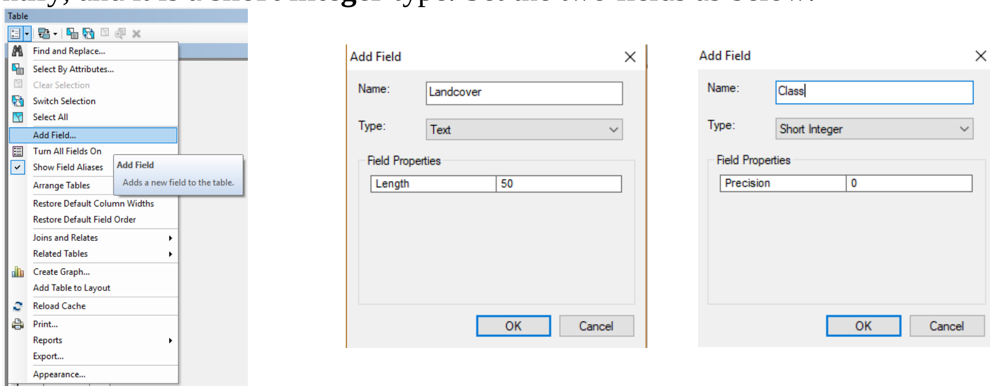
Make the **false-color Landsat** image on the top of all raster images for the best visual effect. Put points on the top of the water in the Landsat image. At least ten points need to be selected. Same as the training data for supervised classification, the reference points need to be evenly distributed among the entire image, covering a wide range of water (fresh and saline water). You can change the color and the size of the points for better examining the distribution of them. To change the size and color of the points, in the table of contents, click on the symbol of the layer GroudTruth. After finishing selecting the water points, in the **Editor** toolbar, choose **Save Edits** and **Stop Editing**.



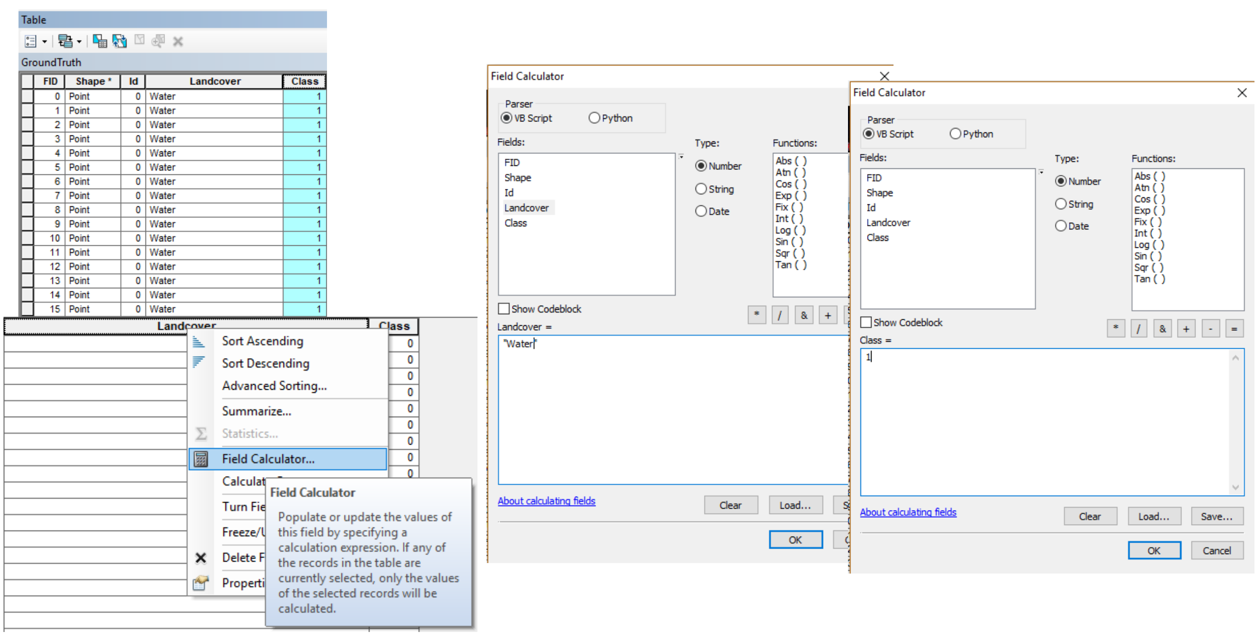


5. Editing the attribute table of layer GroundTruth.

In the **Table of Contents,** right-click the layer **GroundTruth**, choose **Open Attribute Table**. We need to create two other columns to give the identity of the selected points. In the attribute table, click the **Table Options** icon . Choose **Add Field**. The first field is the **Land cover** is the **Name** , and select **text for the** type field. The second field is **Class** identify, and it is a **short integer** type. Set the two fields as below.

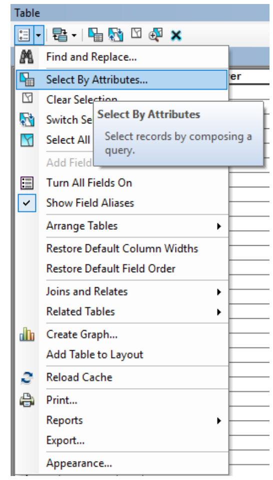


6. Add value to the **Landcover field**. Right click on the title of the **Landcover column**. Choose **Field calculator**. Add the newly selected point land cover type as "water". Since Land cover is string type attribute, you need add quote to the text when adding it in field calculator. Type "**Water**" in the **Field Calculator** as follow. Water is added to the Landcover column of the selected points. Similarly, the class of the water should be given as 1. **To make sure the attribute table can be edited with Field Calculator, you need Save edits and Stop Editing, every time you finish editing one type of ground truth points.**

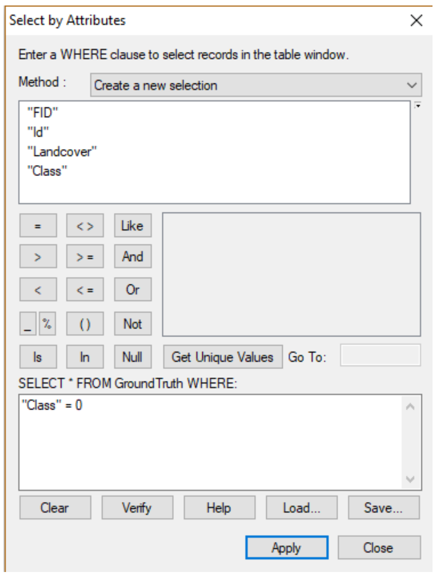


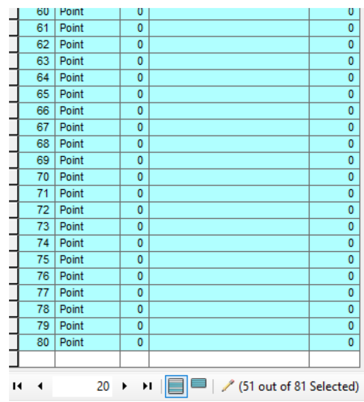
7. Repeat step 1 ~ 3 to select reference points of the second class. Take **Conifers** as an example—selected reference points located on the dark red vegetation area with the rough texture.

Editing the attribute table of the newly selected points is a little different from the first class. Open the attribute table again. You will see the class column of the newly selected conifer points is 0. Let us only work on the conifer points. In the attribute table, click , choose **Select By Attributes**.



In the **Select by Attributes window**, type selection criterion: **"Class" = 0.** In the attribute table, items with class value 0 are highlighted. Click to make only selected items are shown. Follow step 3 to make the **Landcover type as Conifers and Class as 2.**



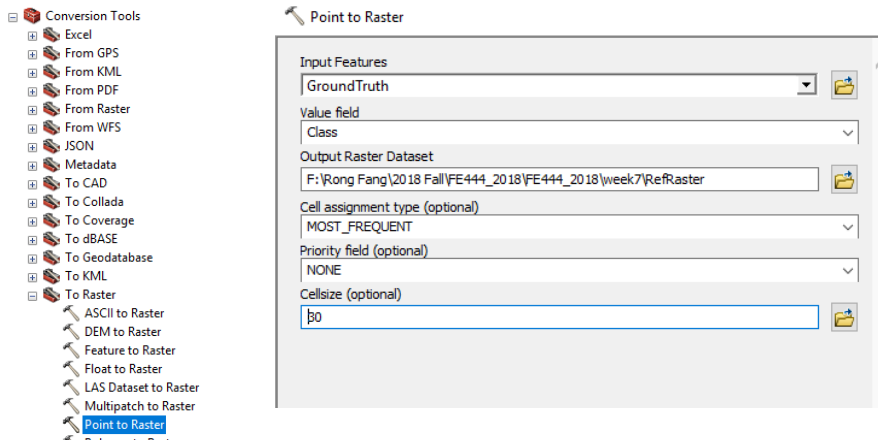


Repeat step 1 ~ 4 to finish selecting reference points for other classes according to your classification. In my case, the six classes are **Water, Conifers, Broadleaf, Pasture, Developed, and Bare land.**

8.Convert the shapefile to raster.

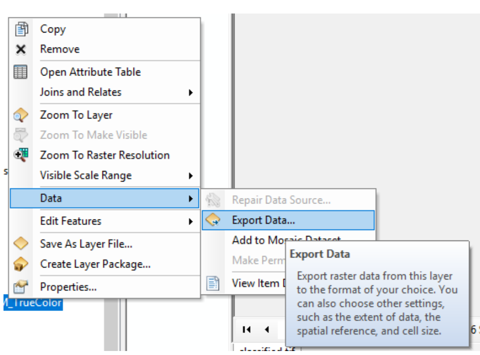
To compare the ground truth layer to the classified image, you need to convert the shapefile to a raster file. Before conversion, make sure the attribute table **GroundTruth point** is deselected. Open the attribute table of GroundTruth, click to clear the selection.

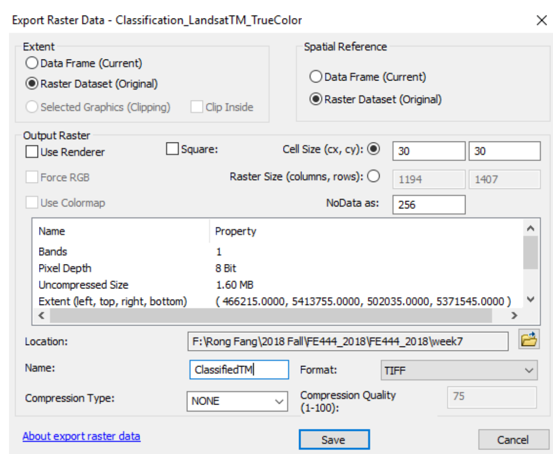
Open **Toolbox** , choose **Conversion Tools → To Raster → Point to Raster.** Type the parameters in Point to Raster as below. Set **Class** to **Value field.** Save the output raster file under the folder Week 6, and name it as **RefRaster.tif. Make sure the Cellsize is 30,** which is consistent with the Landsat TM image. The raster layer will be automatically added to ArcMap.



9.Combine the reference layer and the classified layer.

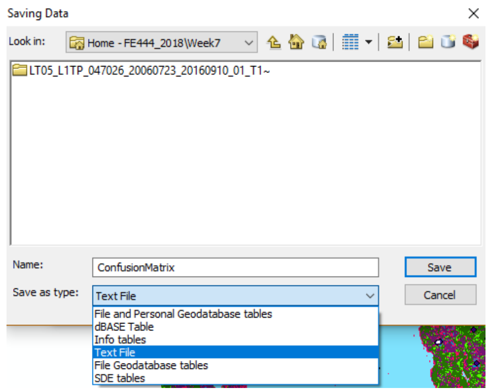
Export the classified layer. Right-click the classified layer, choose **Data → Export Data**. Name the file as ***ClassifiedTM.tif*** , and save it in folder Week 6. Select yes to add the exported file as a new layer.





Next, we need to use **Spatial Analyst Tools** in the **Toolbox**. So make sure your Spatial Analyst Tools is enabled. In the main menu, choose **Customize → Extensions**, and check the box of Spatial Analyst. Open Toolbox , navigate to **Spatial Analyst Tools → Local → Combine**. Choose the ***ClassifiedTM***and ***RefRaster*** as the input rasters. Save the output raster at the Week 6 folder. Name it as **Combined**. The layer Combined will be automatically added to the ArcMap. Right-click the layer **Combined**, and open the attribute table.

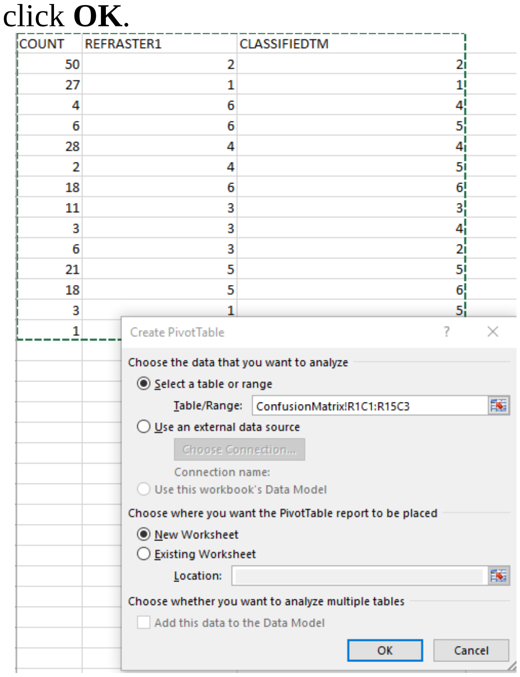
Next, we need to export the attribute table and work it in Excel. In the attribute table, click and select **Export**. Name the exported table as ConfusionMatrix and save it as .txt type at the folder Week 6.



10. Create confusion table in Excel.

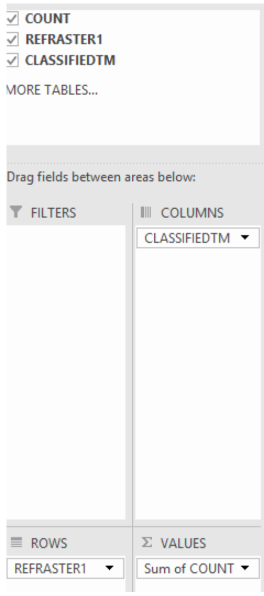
Find the file **ConfusionMatrix** in the folder Week 6. And change the extension name of the **ConfusionMatrix** from **.txt** to **.csv**. Open Excel and open the file **ConfusionMatrix** from folder Week 6 by using File/Open. Delete the column Rowid and Value. Select all the columns and choose Insert in the main menu. Choose PivotTable 

. In the Create PivotTable window, click **OK**.



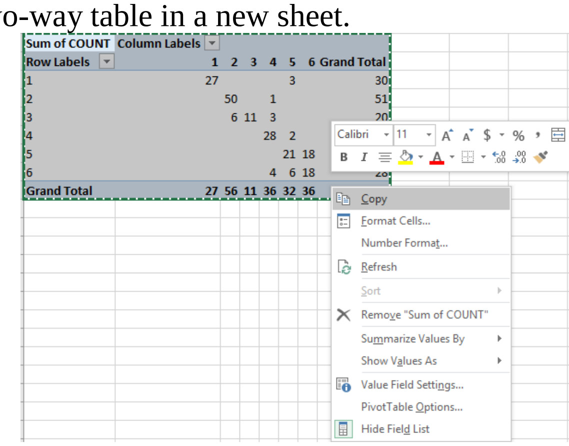
COUNT	REFRASTER1	CLASSIFIEDTM
50	2	2
27	1	1
4	6	4
6	6	5
28	4	4
2	4	5
18	6	6
11	3	3
3	3	4
6	3	2
21	5	5
18	5	6
3	1	5
1		

In the Pivot Table sheet, you will see the Pivot Table Fields. **Choose or drag Rows: RefRaster, Columns: ClassifiedTM, and Values: Count.** The two-way table will be created as below.



Sum of COUNT	Column Labels						
Row Labels	1	2	3	4	5	6	Grand Total
1	27				3		30
2		50		1			51
3		6	11	3			20
4				28	2		30
5					21	18	39
6				4	6	18	28
Grand Total	27	56	11	36	32	36	198

Copy and paste the two-way table in a new sheet.



Change the titles of the classes to their land cover types.

Sum of COUNT	Classified						
Ground Truth	Water	Conifers	Broadleaf	Pasture	Develope	Bare Land	Grand Total
Water	27				3		30
Conifers		50		1			51
Broadleaf		6	11	3			20
Pasture				28	2		30
Developed					21	18	39
Bare Land				4	6	18	28
Grand Total	27	56	11	36	32	36	198

In the two-way table, the count in the diagnosis is the number of pixels that are correctly classified. Let's take class **Conifers** as an example. In the Ground Truth sample, there are 51 pixels selected as the Conifers class. However, 50 of the 51 conifer pixels are correctly classified as conifers in the classified image, and 1 of them are classified as pasture. Totally, 56 pixels of the ground truth raster are classified as Conifers. However, only 50 of the classified conifer pixels are correctly classified in the right class.

The **omission error** of Conifers classification:
(1/51) * 100% ≈ 2%.

That can be explained as classification fails to classify 2% of the reference conifer sample to the right class.

The **commission error** of the Conifers classification:
(6/56) * 100% ≈ 11%.
That can be explained as 11% of the classified conifer pixels are wrongly classified.

Let's update the confusion matrix. The table below shows the red parts are wrongly classified, and the green parts are correctly classified. The yellow parts are the total number of pixels classified to a certain class. The blue parts are the number of pixels that belong to each ground-truth class. Commission error is calculated as the sum of red number in each column divided by the yellow parts (sum of the pixels in the classified raster). Omission error is calculated as the sum of the red number in each row divided by the blue parts (sum of the pixels in the ground truth reference raster).

Sum of COUNT	Classified							
Ground Truth	Water	Conifers	Broadleaf	Pasture	Developed	Bare Land	Grand Total	Ommision Error
Water	27				3		30	0.10
Conifers		50		1			51	0.02
Broadleaf		6	11	3			20	0.45
Pasture				28	2		30	0.07
Developed					21	18	39	0.46
Bare Land				4	6	18	28	0.36
Grand Total	27	56	11	36	32	36	198	
Commision Error	0.00	0.11	0.00	0.22	0.34	0.50		

Q10 (10%). What are omission error and commission error? Provide a screenshots of your final confusion matrix.

6.5 Advanced topics: Image Classification with ERDAS

Lab developed by Sudeera Wickramarathna and Dr. Strimbu from the Dr. Hilker and ERDAS documentation.

Note: throughout this lab, the terms class, category, and land-cover type can be taken to mean the same thing and are used interchangeably.

6.5.1 Objectives

- Exploring spectral properties with 2D Scatter Plots
- Execute supervised classification of images
- Define areas of interest (ROI)

6.5.2 Files to be used in the Lab

- LandsatTM is a Landsat 5 image of the Gulf Islands from south BC acquired in 2006.

6.5.3 Supervised Classification

Supervised classification of remotely sensed data begins with delineating boundaries around groups of spectrally similar pixels, known as "training areas", which are representative of specific land-cover types. The location of training areas might be determined in the field using GPS; interpreted directly from other forms of remotely sensed data (e.g., aerial photography); or other sources available for GIS software (i.e., existing digital maps). Training areas (i.e., groups of spectrally similar contiguous pixels) representing specific land-cover classes, once delineated, are used to calculate statistics characterizing the distribution of pixels' Digital Numbers (DN) in all bands/channels representing each particular land-cover type. Some of these statistics include: Minimum, Maximum, Mean, Std. Dev. These statistics are used to assign membership to a class/category (land-cover type) for each image pixel by applying an image classification algorithm (e.g., Maximum Likelihood).

In summary: groups of spectrally similar contiguous pixels are delineated and act as training areas representing specific land-cover types/classes/categories. The digital numbers/values associated with these training areas are then statistically summarized. These statistical summaries facilitate assigning membership to all other pixels in an image. Membership for each individual pixel is assigned based on the values representing a particular class closest to the values contained in an unclassified pixel. The end result is a classified image displayed with colors assigned to represent each class/category/land-cover type.

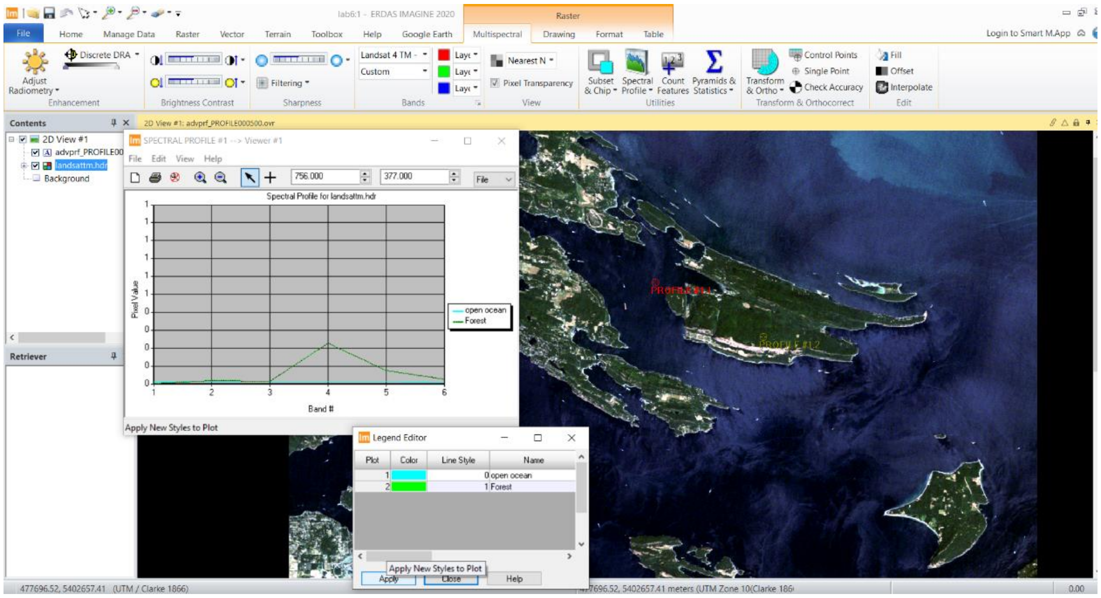
To familiarize your self ith the image, open the *LandsatTM* file in your main View. This is a layer stack representing 6 bands/channels of Landsat TM imagery acquired on July 23, 2006; wherein: b1 (blue); b2 (green); b3 (red); b4 (near-infrared (NIR)); b5 (shortwave infrared (SWIR)) and b6 (SWIR). **NOTE**: Band 6 in this layer stack is actually Landsat Band 7 (SWIR). From the available bands' list, display an RGB color composite, using bands 1 (blue), 2 (green), and 3 (red). This will display a true color composite. In the **Main Image View**, **Adjust Radiometry, choose *Standard Deviation Stretch*.**

6.5.4 Exploring spectral properties of land-cover features using Spectral profiles

The **Spectral Profile Viewer** allows you to visualize the reflectance spectrum of a single pixel. If the image is hyperspectral, we can see the reflectance spectrum of each band using the **Spectral Profile Option**. Similarly, the spectral profile function can be applied to see different features (i.e., water, urban areas, forested areas, etc.) that reflect different wavelengths.

1. Left-click on the "**landsattm.hdr"** on the Content pane. Now you see there are four other tabs under the **Raster tab.** Select **Multispectral Tab** and go to the **Utility group**. Select the second option from the left (**Spectral profile**) by clicking the drop-down menu.

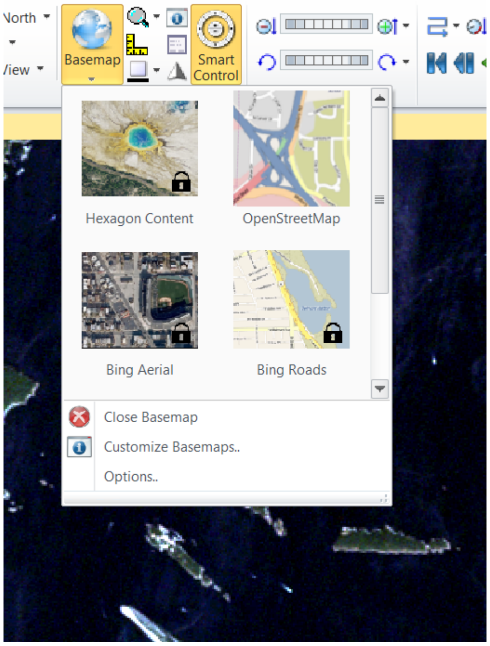
2. A separate window will appear, named **Spectral Profile.**



3. From the new window, **select +** icon and click one of the pixels on the image where you are interested to see the spectral profile.
4. You can continue this step by selecting different landcover types in the image. Now you see how the **Pixel values** are varying with bands.
5. Next, click Edit on the **Spectral Profile** window and select the **Chart option**. Now you can change the Title, **X-axis, and Y-axis** attributes accordingly.
6. Similarly, select **Chart Legend** and change the **Color, Line Style, and Name** of each profile.

Q#1(10%). Include a screenshot of spectral profiles containing pixels from a forest, waterbody, sandy beach, and urban areas.
Add proper legend with different colors to see the spectral profiles. Also, Change the line type appropriately.

NOTE: you can insert a base map underneath need the Satellite image ("landsattm.hdr") by **Home>Basemap (View group)** > **OpenStreetMap**. This will help you to identify landcover features (e.g., Urban Areas). Also, you can use the **Google Earth** option as well to identify landcover features.

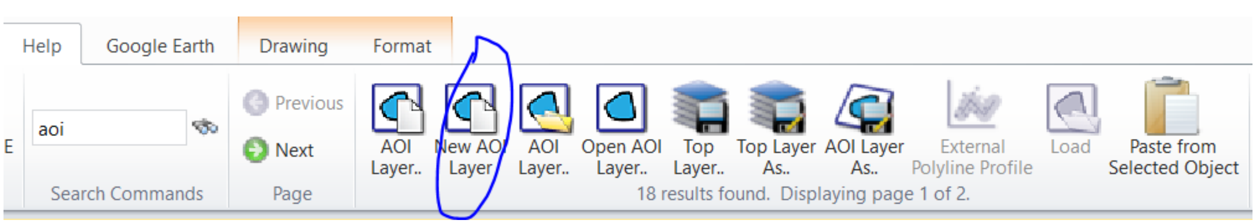


6.5.5 Exploring spectral properties of land-cover features using 2D Scatter Plots

A **feature space image** is simply a graph of the data file values of one band of data against the values of another band (often called a scatterplot). The image can be either unsigned 8-bit, 16-bit, or 32-bit. A feature space image can be used in any of the utilities, for example, the 2D View, Classification, Map View, Spatial Modeler, and so forth).
When you display a feature space image file (*.fsp.hdr) in a Viewer, the colors reflect the density of points for both bands.

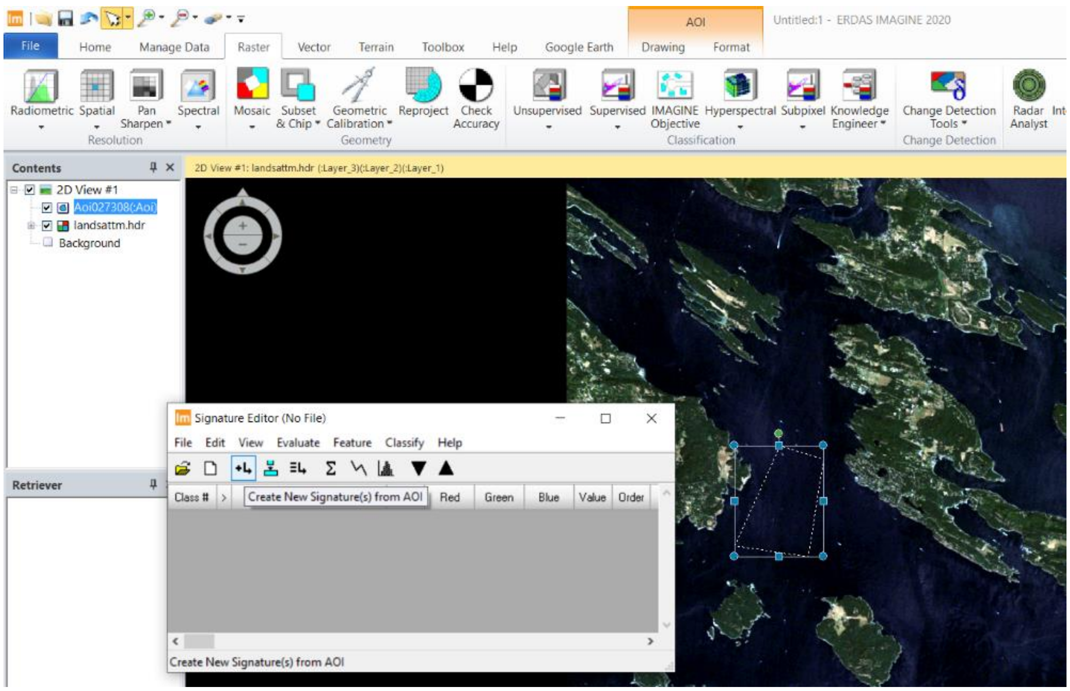
- Bright tones represent a high density
- Dark tones represent a low density

1. Remove previous images from the main View and load **"Landsattm.hdr"** again to the main View.
2. Set the **Landsattm** image to the True color image.
3. Go to the **Help tab** and type **AOI under** the search command. You will see the related findings next to the **Search Command group**. Select the **New AOI layer** option.

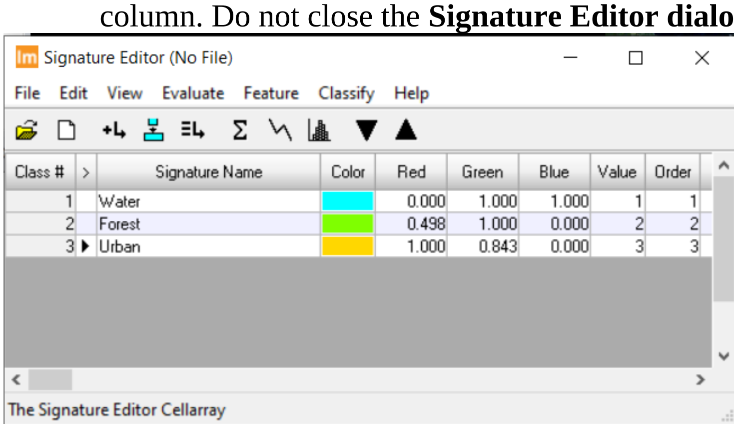


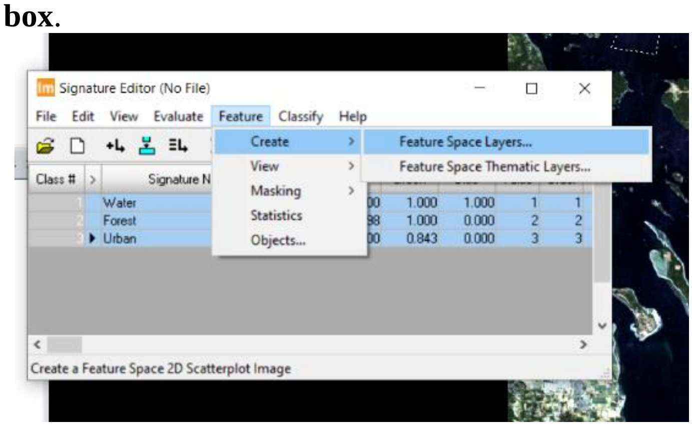
4. Now you will see that the AOI layer has been added to the **Content pane**. Always make the **AOI layer** as the top item in the **Content pane list**.
5. Go to the **Raster tab** and click the **Supervised** drop-down menu under the classification group. Then **select Signature editor** option.

6. A **Signature Editor dialog box** will open as a **separate window**.
7. Now, Click on the **AOI layer** in the **Content Pane**. Now you will see the **AOI tab** has activated. Go to **AOI | Drawing** and click **polygon icon** from Insert **Geometry group**.
8. Identify distinct landcover types from the true color image and draw a polygon covering that land cover type (e.g., Select Water)
9. Go to the **Signature Editor dialog box** and click the **Create New signature icon**.



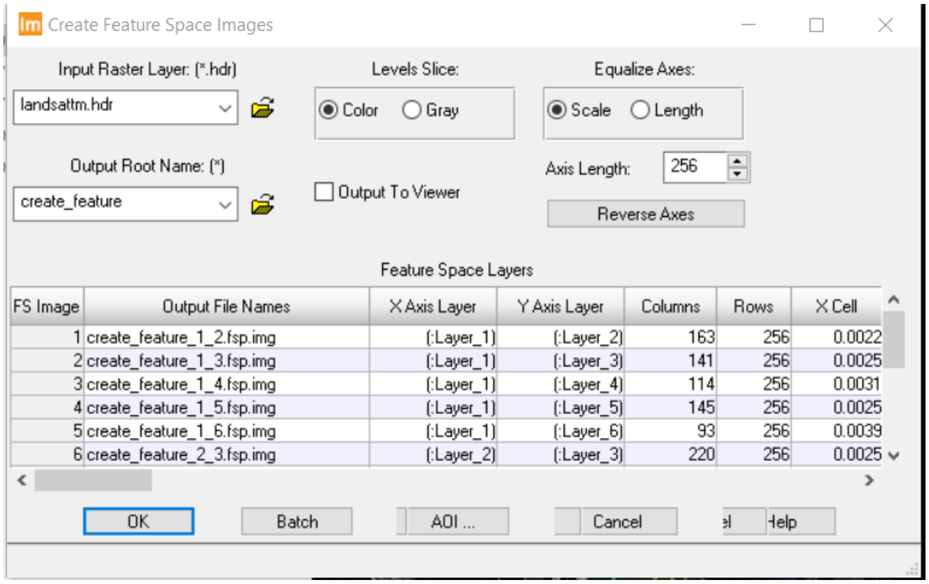
10. Now you can see the polygon has added to the **Signature Editor Table**. **Click Class 1** under the **Signature Name column and rename it as Water**.
11. Continue the above steps and add two more land cover types (i.e., Forest and Urban) to the **Signature editor dialog box**.
12. Change the color of each signature appropriately using the drop-down menu in the Color column. Do not close the **Signature Editor dialog box**.





13. Select the three signature columns and select **Feature** > **Create** > **Feature Space Layers**.

14. Then **Create Feature Space Images** dialog box will open. Navigate to the data folder and select "**landsattm.hdr"** image for **Input Raster Layer**. **Type Create_feature** for the **Output Root Name**. Keep Other settings as default.

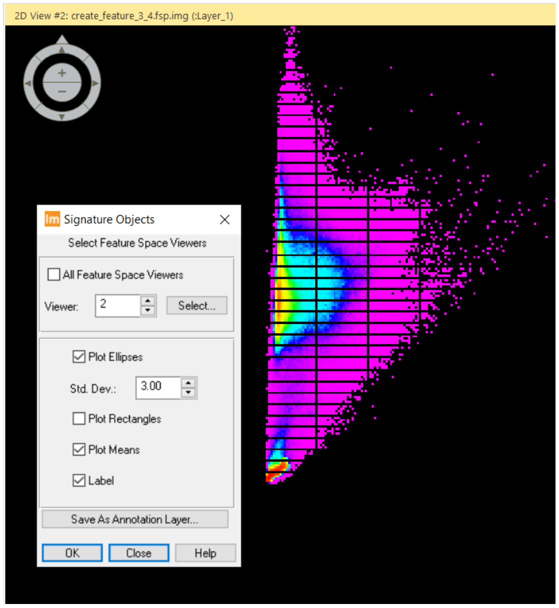


15. **Click OK**. Once the **Process List** has been completed, click **Close.**
16. Now go to **Home Tab** and **select Add View**. Then **Select Display Two Views**.

NOTE: **Feature Space Images** provide a graphical view of all x, y combinations associated with the spectral values of two channels of remotely sensed data. To explore this utility, we will graph two bands that exhibit very different spectral characteristics for the most common component of this image (i.e., vegetation). Landsat TM Band 3 measures reflectance in the red (visible) portion of the spectrum, while Band 4 measures reflectance in the NIR portion of the spectrum. Typically, healthy vegetation exhibits very low levels of reflectance in the red vs. very high levels of reflectance in the NIR. The area between this low and high reflectance associated with the transition from red to NIR is known as the "red edge".

17. Navigate to your output folder where you saved the **Feature Space Images**. Select create_feature_3_4. Note that the xxxxxxx_ 3_4 (and other combinations) represent the bands that are being plotted in the image. In this case, layer 3 (band 3) will be displayed on the x-axis and layer 4 (band 4) on the y-axis.

18. **Select File >Open > Raster Layer** and navigate to the data folder and select **"Create feature_3_4.fsp.img"** file that you just created.
19. Now it will add to the **Second View (2D View#2)** and set Fit to Window by right-clicking the View#2.
20. Now go back to the **Signature Editor dialog box. Select Feature >Object. A new Signature Objects dialog box will open.**



21. Change **Viewer** to 2, because your feature layer is in View 2.
22. Check **Plot means and Label**. Click **OK.**
23. Now you will see a **Feature space graph with labels and with prominent landcover areas.**

NOTE: In the feature space plot, red (band 3) is on the X-axis and near-infrared (band 4) is on the Y-axis. You can explain the positions of your landcover classes (Water, Forest, and Urban) in feature space plots as follows:

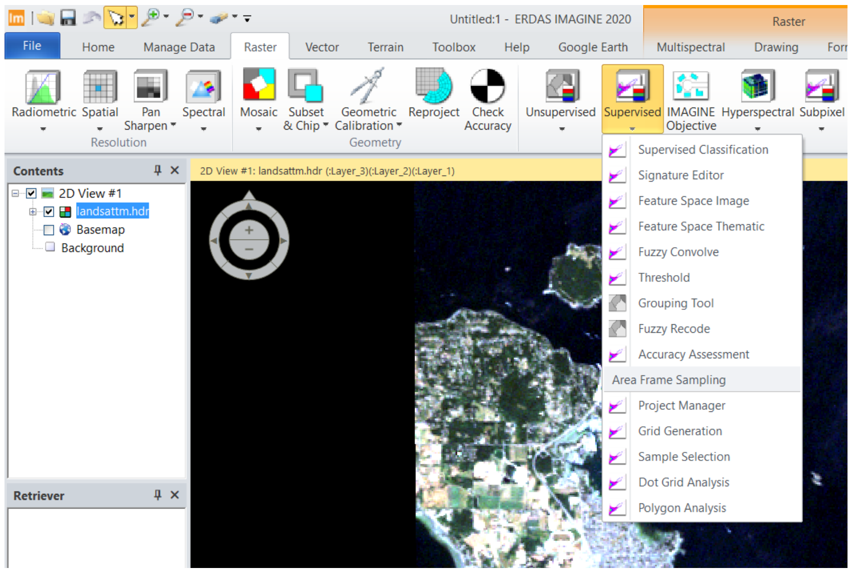
• **Water:** Low values in both band 3 and band 4.
• **Forest:** High values in band 4 and low values in band 3
• **Urban:** High values in both band 3 and band 4

Q#2 (15%). Include a snapshot of your feature space plot for band 3 and band 4.

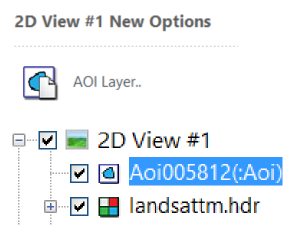
6.5.6 Define areas of training and testing data using Area of Interest (AOIs): Generating Signatures

To perform supervised classification of digital remotely sensed data, training (calibration) data are required to provide examples of the statistical values representing particular land-cover classes. In ERDAS, we delineated individual training sites as **"area of interest (AOI)"** with a particular land cover class. Pixels within the ROI (training site) generate a unique signature, and we use AOIs to manually delineate locations that represent areas of training data.

1. To begin this process, in the **Main Image window**, select **Raster Tab,** and then **Signature Editor** option by clicking the **Supervised** drop-down menu.
2. A signature window will open and expand the window to see all the columns.



3. Now go back to the **File tab and select New**. Select the **AOI Layer** option from **2D View #1 New Options**. A new **AOI layer** will appear in the Contents pane. In the next steps, we are going to use this layer to draw the training polygons.



- We are going to classify the 2006 Landsat TM image by defining AOIs for several land-cover categories. There is no specific rule dictating how many land-cover categories one should aim for; however, bear in mind that ***a land-cover class is only realistic if it is spectrally unique and observable at the spatial resolution of the imagery.***

- In other words, in most cases, spatial and spectral resolution of Landsat imagery allows for distinguishing between taxonomic tree types, such as broadleaf and coniferous, but do not permit separating specific tree species, such as Douglas-fir and Western redcedar.

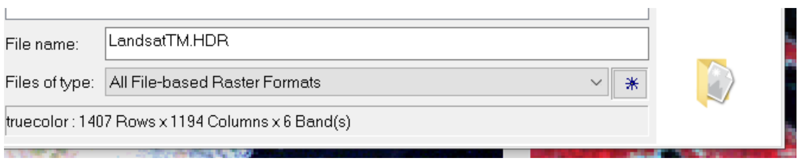
- Therefore, defining broadleaf and conifer as target land-cover categories is practical and achievable, as these are realistic spectral classes, whereas classes based ***on specific tree species (e.g., Douglas-fir and Western redcedar) are unrealistic as these information classes (i.e., species type) are not spectrally distinguishable.***

- When considering Landsat imagery, beyond forests, other types of vegetation can be distinguished. For instance, frequently, areas dominated by herbs, shrubs and/or agricultural growth will appear spectrally different to forests in Landsat imagery; however, these non-forest vegetation types may appear spectrally similar to each other.

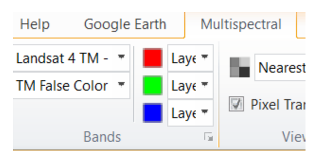
- An exception to this relates to vegetation health, which can be associated with things such as stress (due to water shortage and/or pest infestation). Furthermore, the time of year is important. Due to phenological properties, vegetation may appear very differently in July and January. Beyond vegetation, other types of land-cover features are spectrally identifiable in Landsat imagery. These may include classes relatable to water and/or human-made features.

4. At this point, open a second View **(2D View #2)** by selecting Add **Views >Display Two Views in Home Tab** (you can refer to steps as in Lab 1 → 3.4 Multiple Views). Right-click on the **2D View#2 content pane** and select **"landsattm.hdr."**

   NOTE: If you cannot see the **"landsattm.hdr"** File after you navigate the correct folder, you can select **Files of type** as **"All File-based Raster Formats."** This option will allow you to see all the files in the data folder.



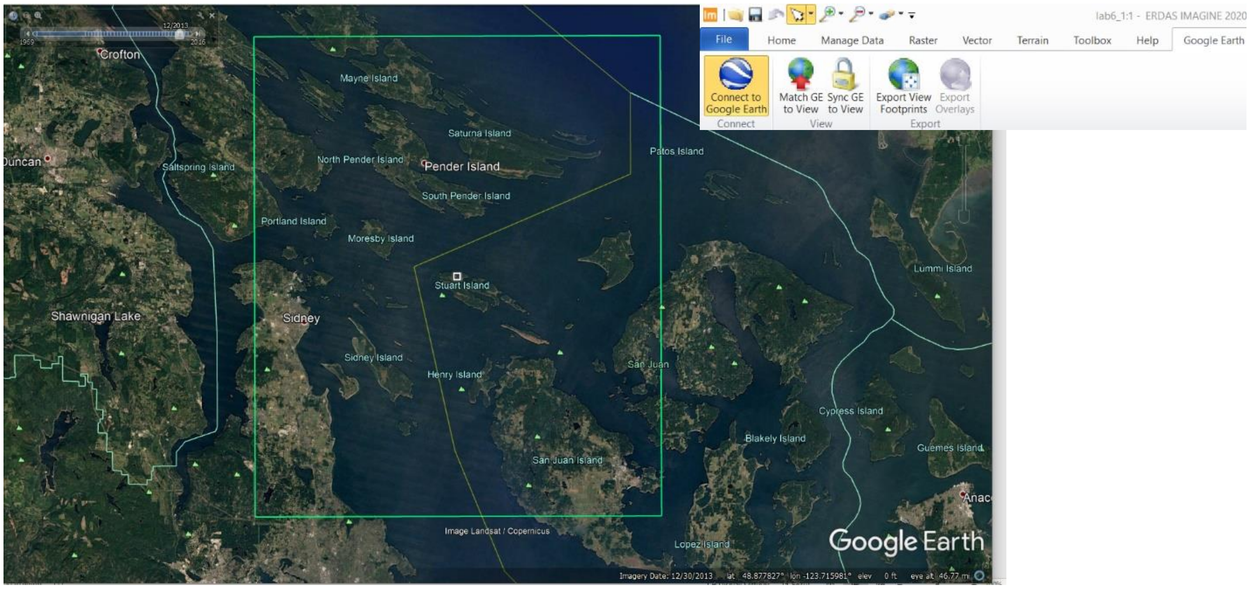
5. Display a false-color RBG composite of red (Layer 4), green (Layer 3), and blue (Layer 2) in the **Bands group** under the **Multispectral Tab.** Similarly, you can select **TM False Color** as the Band combination.



6. Under the **Home tab** select the **Link Views option (next to Add Views option)** to synchronize both the Views.

   NOTE: By adding a second view with a **False color composite** image, you can easily distinguish the landcover features.
   Also, you can insert **Google Earth view by clicking the Connect to Google Earth** option.

Further, you can use other options such as **Match GE to View, Sync GE to View**, and **Export View Footprints to navigate your AOI.** This allows you to see the landcover features very clearly.

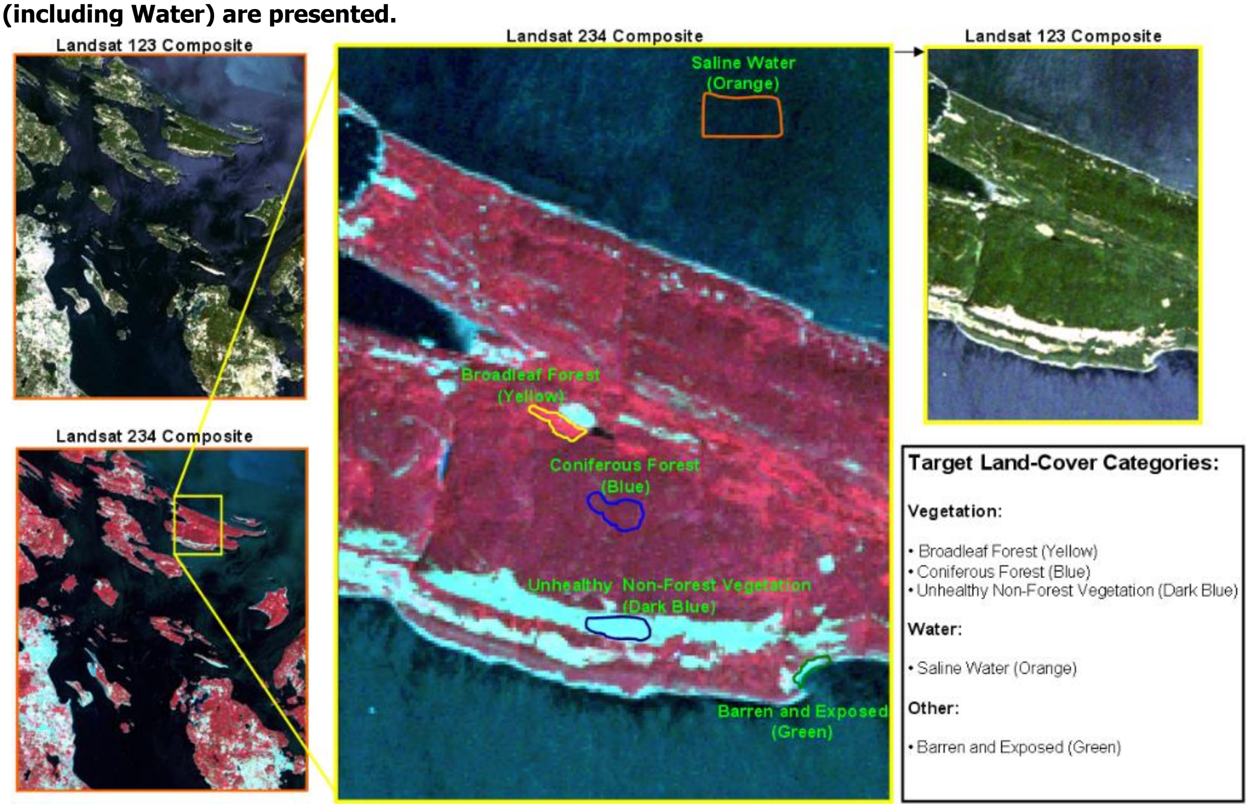


Series of images below and on the following pages show you examples of basic land-cover types present in the Landsat image.

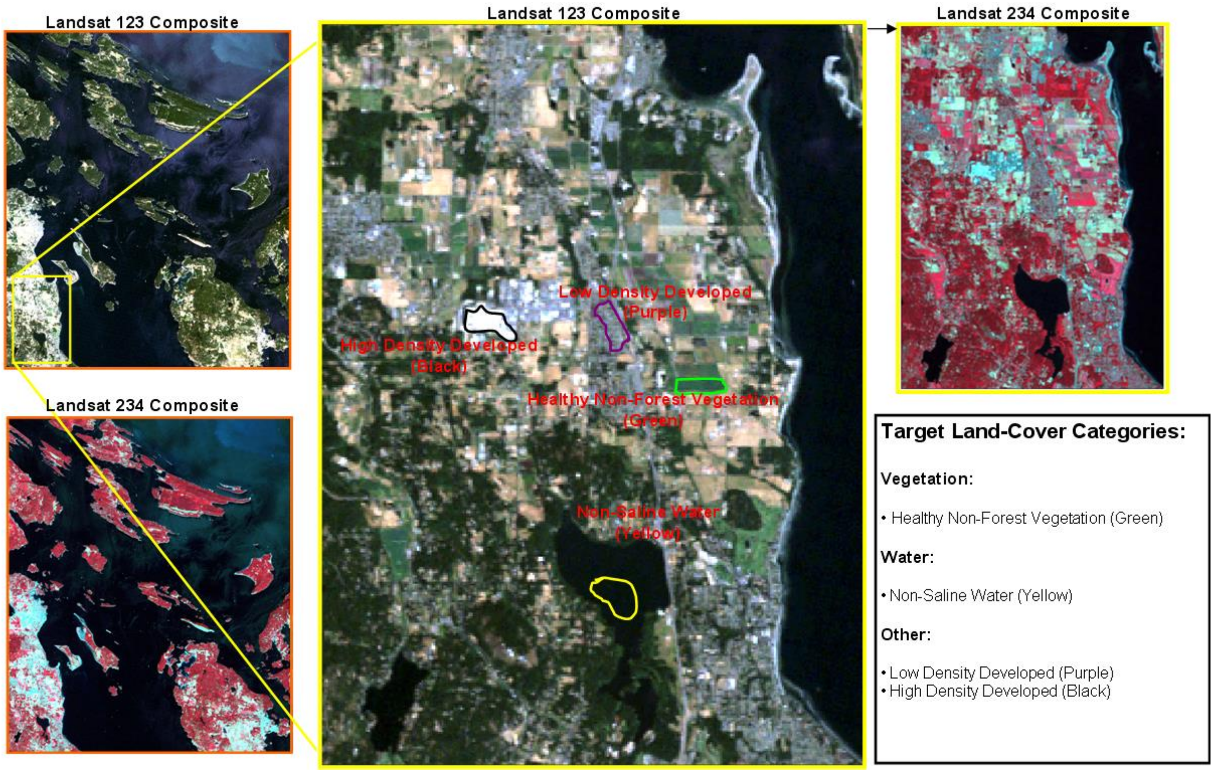
NOTE: The images represent the SAME area and class examples. Use the classes highlighted in the series of images to form the basis for your classification. PLEASE BEAR IN MIND that even though two types of Water are presented (i.e., saline (salt) and non-saline (fresh) Water), they should be treated as ONE land-cover class when deriving calibration and validation data.

- Two are only presented to make a distinction that there are 2 water types; however, they are not spectrally distinct. Therefore, in total, eight classes are presented.
- You are encouraged to locate and delineate additional categories; however, failure to do so will NOT result in a deduction of marks. Eight classes are certainly enough for this introduction's purposes and scope; however, if considering additional classes, they must be spectrally distinct in and of themselves and in relation to other defined categories.
- They must also be land-cover features captured by the spatial resolution inherent to the imagery (30 x 30 m pixels), and the time of year of the imagery must be considered (if applicable to a particular class).

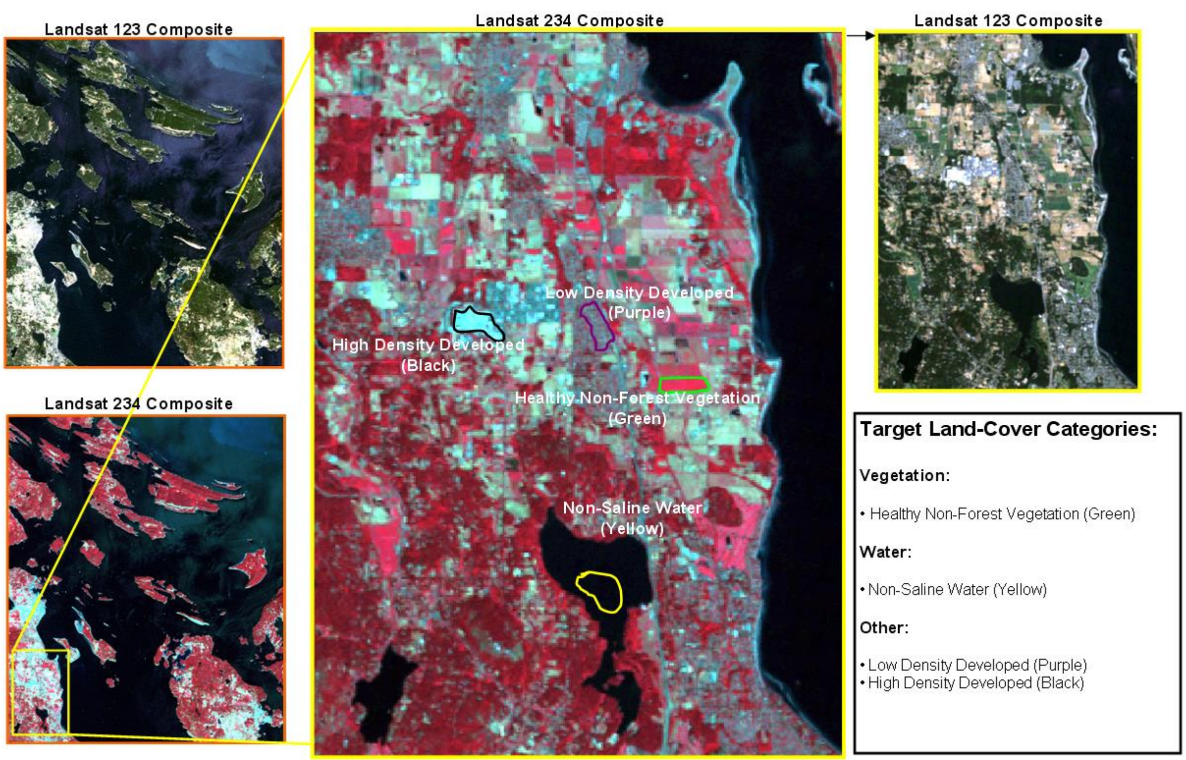
Location of basic land-cover classes focusing on Landsat TM true color (123) composite. Five classes (including Water) are presented.



Location of basic land-cover classes focusing on Landsat TM false color (234) composite. Five classes (including Water) are presented.



Location of basic land-cover classes focusing on Landsat TM true color (123) composite. Four classes (including Water) are presented.



Location of basic land-cover classes focusing on Landsat TM false color (234) composite. Four classes (including Water) are presented.

7. To delineate AOIs for a specific land-cover category (e.g., broadleaf forest), navigate to an area of interest (that you believe in representing a specific land-cover category) and select the **Polygon icon** in located in **Insert Geometry** group **Under Raster | Drawing Tab**.
   NOTE: Bear in mind that you have to draw the polygons or select the training sites on a true-color image (i.e., 2D View#1). It is always better to Scroll and Zoom the image and draw the polygon to have a better view of the landcover feature.

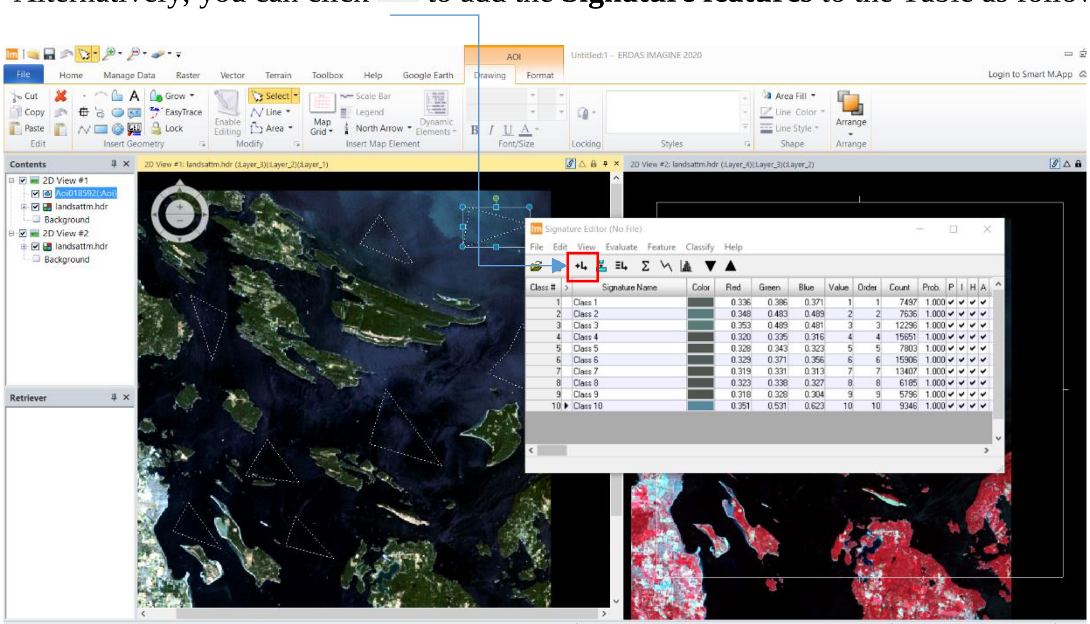
   In other words: bearing in mind which View your **Drawing tool** is associated with, use the other displays (from which you did not use to draw polygons) to navigate to an area of interest, then turn the **Polygon tool** on and proceed to delineate a polygon around the perimeter of a group of pixels representing the land-cover Feature of interest (i.e., specific to a land-cover category) using the left mouse button.

8. Once you have outlined the shape/perimeter of a feature/area of interest, two right-clicks will close the polygon (the first right click closes the polygon and the second right-click accepts the polygon). If you are unhappy with the polygon that you have just completed, at any point, you can navigate back to it and place the cursor over it, and hit the middle mouse button, and delete the polygon.

SAVE YOUR FILE OFTEN! Choose *File,* Sessions, and *Select .ixs*. The file should be saved as calibration.ixs.

9. Make sure that you have activated the **AOI layer** all the time. Now you can add a similar type of land cover features to the **Signature Editor**. For example, we can first collect all the **Signature Files relevant to Water** and then move to Forest signature files.
   Once you have completed each polygon, go to the **Signature Editor dialog** box and **select Add.**

   Alternatively, you can click to add the **Signature features** to the Table as follows.



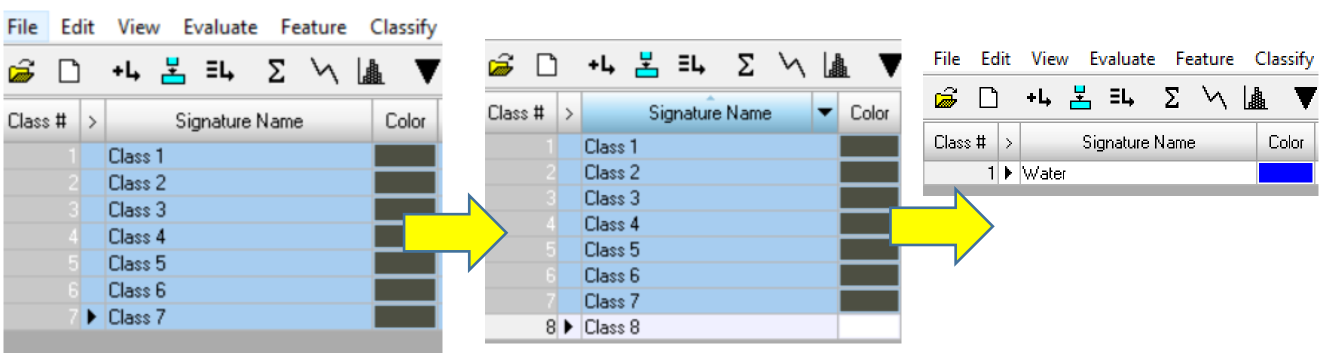
Q#3 (15%). Include a snapshot of your main window (similar to the above figure) that includes your AOI polygons and the false color image. Also, add the snapshot of the Signature editor dialog box as well.

NOTE: To add additional polygons to the AOI representing a specific land-cover category (e.g., broadleaf forest) continue to navigate to areas of interest in the image and delineate the shape/perimeter of additional groups of pixels representing the land-cover Feature of interest, solidifying these additional polygons with two clicks.

• For every land-cover class, you need multiple polygons. It is better (and required!) to have numerous small class-specific training areas spread out throughout the image than to have only a couple of very large training areas.

• To capture class variability, you should have at least 10 different polygons per class as evenly spread throughout the image as possible.

10. Once you have collected an appropriate number of polygons, click the **Signature Names + Ctrl key to select the rows.** Then click the **Merge Selected Signatures** icon . This will add a new row by combining the **Spectral Signatures of polygons** that you have collected. Now right click on the highlighted rows and delete them. Now you will see only the combined class, click it and rename it (e.g., **Water).**



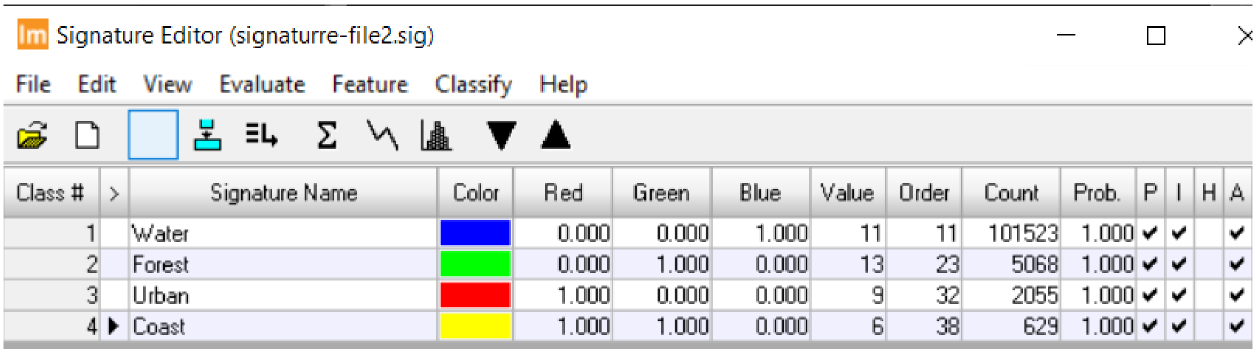
• For Water and vegetation classes, you are REQUIRED to have 10 polygons per class.

• For all other classes (i.e., barren and exposed; low density developed; high density developed; and any additional classes you may have defined), you are required to have 10 polygons per class.

NOTE: To delineate **AOIs** for additional land-cover categories (e.g., conifer forest), select appropriate areas in the main window (2D View#1). You can then follow the same process just outlined to delineate class-specific polygons for each class.

Color is used to distinguish between different classes. ERDAS picks a default color and name for every new region. To change the color of the new region, right-click on the ***Color* field**, whereas you can change the name of a region by double left-clicking in the **Signature Name field** and typing the desired category name (e.g., coniferous forest).

11. Once you have established eight or more classes, each represented by an AOI composed of multiple polygons (i.e., 10 depending on class), click **File > Save As** in the **Signature editor dialog box**. Navigate to your Lab 6 data folder and name as **"signature_file,"** and click **OK**. Count column containing an adequate number of pixels (i.e., > 100-500 depending on class), save *signatrure_file* one final time.



12. Also, close the **Google Earth Map** window and **Signature Editor** window as well.

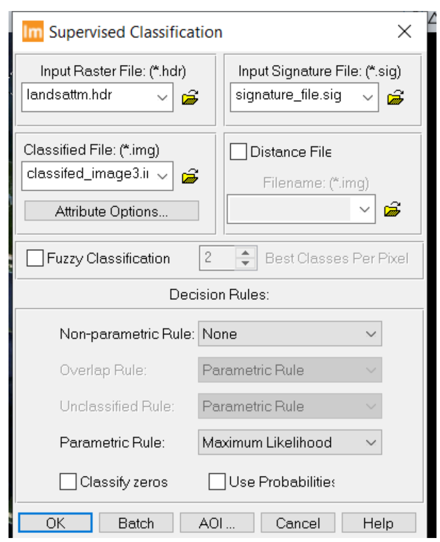
Q#4 (15%). Include a snapshot of your final Signature Editor dialog box.

3.1 Image Classification

We will then classify the Landsat TM image using the **Maximum Likelihood** classification algorithm and all 6 bands as input.

Maximum Likelihood: This classification algorithm assumes that the statistics for each class in each band are normally distributed. This approach also calculates the probability that a given pixel belongs to a specific class that we have introduced during collecting trading data. Therefore, each pixel in an image is assigned to the class that has the highest probability.

1. From the **Raster Tab,** choose **Supervised Tab** and select the **Supervised Classification** option from the drop-down list.
2. Next, the **Supervised Classification dialog box** will appear. From that box, select **"landsattm.hdr"** as the **input raster file** and select "**signature_file.sig"** as the input signature file. You can name the **Classified File** as **"classified_image.img"** and turn on the **Distance File** option, and name it as **"distance_file"** in the same data folder. This **"distance_file"** need for setting the threshold values for the classification.



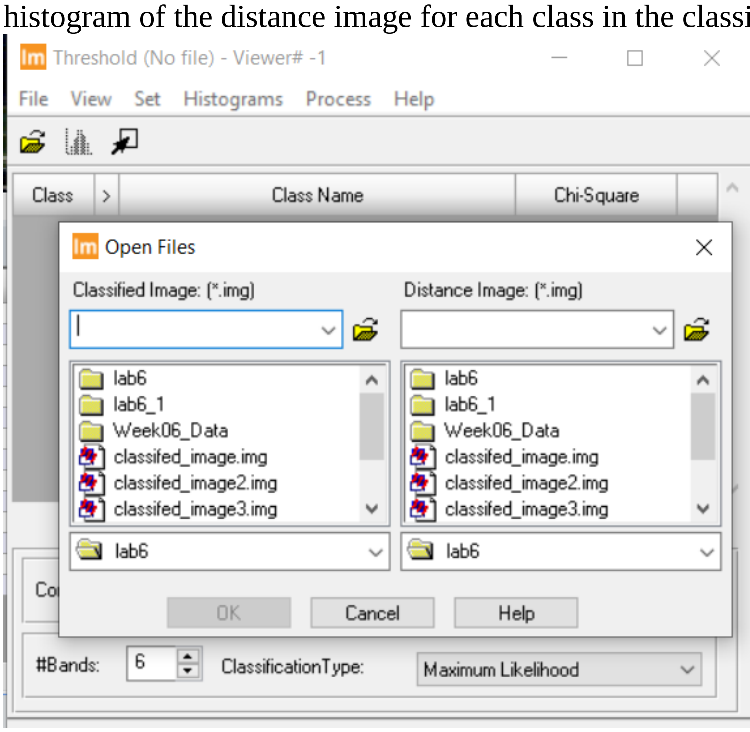
NOTE: Basically, the **Thresholding utility** allows you to determine pixels that are most likely classified incorrectly. Thresholding in supervised classification requires a distance file. (by setting the distance threshold pixels "data distance" from the class mean)

This means that if the highest likelihood calculated for all AOIs (i.e., all classes) for a given pixel is less than 0.95, then the pixel will remain unclassified

3. Then select the **Maximum Likelihood** form the **Parametric Rule** drop-down menu. Click **OK** to proceed.
4. Right-click on **2D View#2** and add the classified image you just created. Compare the classified image vs. satellite image by using options available in the **Window group in the Home Tab.**

Thresholding

5. Select the **Threshold option** from the classification menu under **Raster Tab,** and you will see the **Threshold dialog box**. Select **#Band as 6 and Classification Type as Maximum Likelihood.**
6. From that dialog box, select **File > Open**. Select both the **classified image and distance images** that you just created from previous work. **Click OK**.
7. Then go to **View > Select Viewer** in the **Threshold dialog box** and click on the Viewer **(i.e., 2D View#2),** which displays the supervised classified image.
8. From the Threshold dialog, select **Histograms > Compute**. Now you will see a separate histogram of the distance image for each class in the classified image is computed.



9. Now select each class in the **Threshold dialog box** and click the **Histogram icon** ,a new window will open with a **histogram for a class** (i.e., Water)
10. You can select the **arrow on the X-axis** of the histogram and move to the position where you want to set up a threshold in the histogram (the chi-square value will automatically update as you change the X-axis: try to adjust the Chi-sq. ~11.07 where p=.05 for 6 spectral bands)
    Source: https://www.mathsisfun.com/data/chi-square-table.html
11. Continue the above steps for other classes as well.

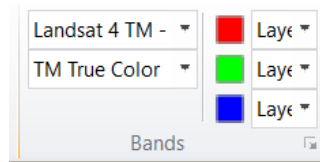
12. Close in the **Histogram window,** and select **View > View Colors > Default Colors** in the **Threshold dialog box**.
13. Select **process> to File** in the Threshold dialog box and save output image as **"threshold_image"** into the Lab 6 data folder.
14. Now you can create a new view (i.e., 2D View#2) and add the **"threshold.img"** file to it.

Q#5 (15%). Include a snapshot of your main window that includes your "threshold.img".

AS ALWAYS, ensure you know WHERE you are saving your files.

Accuracy Assessment

1. Click the **Raster Tab** and select the **Supervised drop-down menu under Classification Group.**

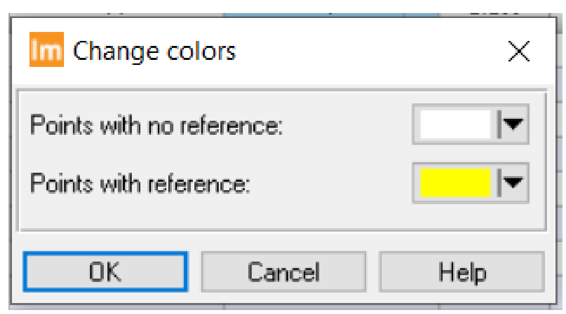
   Don't forget to load your **"Landsattm.hdr"** image to the **2D View#1** and set up it to True Color using **Bands Group** under **Raster | Multispectral Tab.**

2. Select **Accuracy Assessment** and open it. From the accuracy assessment dialog box, select **File > Open.** Then the **Classified Image dialog box** will open. You can navigate to your Lab 6 data folder and select the classified image you saved before. **Click OK,** and the Accuracy Assessment dialog box for the classified image will be loaded.

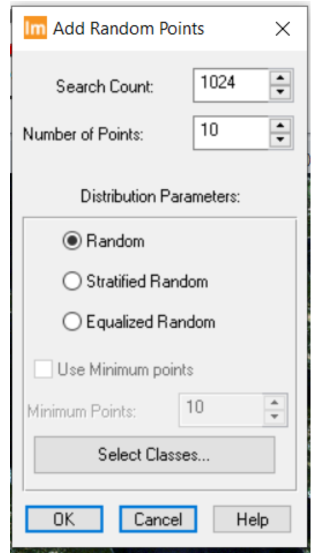
3. You will use this for accuracy assessment.

4. Now **Select View > Select Viewer** From the accuracy assessment viewer. Form the **Viewer Selection Instructions** dialog box and select the classified image.

5. Next, you can choose colors for reference pixels. In the **Accuracy Assessment Viewer, select View > Change Colors.** Here, you can select White for Points **With No Reference** (These points are just random points that have not been assigned a reference value) And Yellow for **Points With Reference (**These points are the random points the have been assigned a reference class value). Click OK. If you want, you can change these default colors.

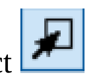
NOTE: The option Create/Add Random Points available under Edit in the Accuracy Assessment dialog box will generate random points throughout the classified image. Once these points are generated, you must enter the class values for the points (reference points). Then these reference values will be compared to the class values of the classified image.

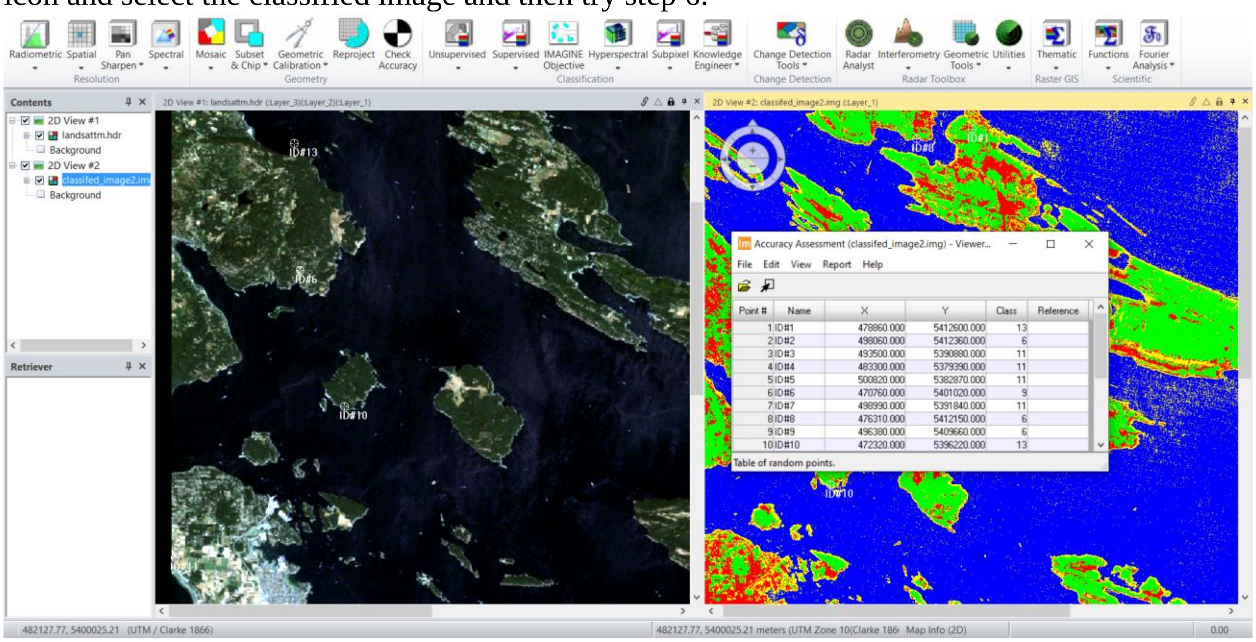


1. Select **Edit > Create/Add Random Points** in the accuracy assessment viewer. Then the **Add Random points dialog** box will open.
2. Here you can set **Search Count as 1024**. This value allows 1024 points to be analyzed to see if they meet the defined requirements in the dialog box. You can change this number if you want but keep the default valve 1024 for now.
3. **Change the Number of Points to 30.** This allows you to generate random points over the classified image. However, to perform a better accuracy assessment, you may need more than 250 points.

4. Under the Distribution parameters options, you have three options, including **"random", "stratified random", and "equalized random".** You can select the **Random option** for this exercise. Then click OK to generate the points.

5. Now you will see a list of points in the accuracy assessment dialog box.
6. In the **Accuracy Assessment dialog box, select View > Show All.** This will show you all the random points that are distributed in your classified image.

NOTE: If you cannot see the random points in the classified image, you may need to select  icon and select the classified image and then try step 6.



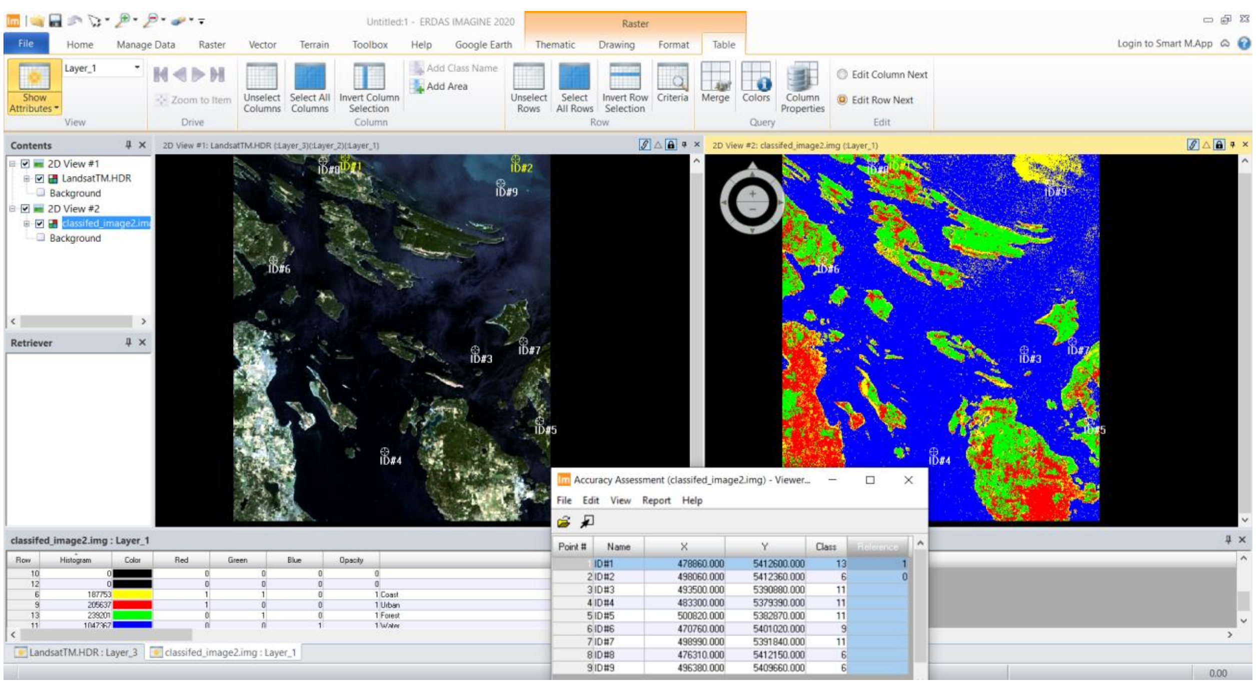
7. Go to **Home and select Equalize sizes**, then select **Link Views and Sync Views.**

Q#6 (15%). Include a snapshot of your main window that includes your true-color image, classified image, and Accuracy assessment table.

NOTE: You will see both images along with their points in white color. Now you can zoom in to a point in the classified image, and the same point will be zoom in the reference image as well. Also, you can add an **Attribute table** related to the classified image by selecting the classified image and **Table> Show Attributes.**

- This Table will show up at the bottom of your main window, and you can use this to find the class number for each class you defined (e.g., Water: class number 11).
- You can do the same task by referring to the signature file you saved before your Lab 6 data folder. Click **Supervised > Signature Editor.**
- Now **Select Open in Signature Editor dialog box and navigate and select your signature file.** Now you will see the class with their respective class values in the **Signature Editor dialog box**. You can refer to these values to fill the reference column in the Accuracy Assessment dialog box.

8. In the **Accuracy Assessment Reference column**, you have to enter the best guess of a class value for the pixel covered by each reference point.
9. Once you have filled the reference value for each point, you can see points in the Viewer will change it to yellow color.



In order to assess the accuracy of the classification, we can use **Error Matrix, Accuracy Totals, and Kappa Statistics available in ERDAS.** You can take this information as a report.

1. Go to **Report > Accuracy Report and Report > Cell Report in the Accuracy Assessment Viewer**. A text file will be created with all the accuracy measurements. If you want to save the reports as a text file.

2. Now close the text editors and go to **File > Save Table in the Accuracy Assessment viewer to save the data.** Once you have finished, close the Accuracy Assessment viewer.

NOTE: The error matrix simply compares the reference points to the classified points. The Kappa coefficient expresses the proportionate reduction in error generated by a classification process compared with the error of a completely random classification. For example, a value of .82 would imply that the classification process was avoiding 82% of the errors that a completely random classification would generate (Congalton, 1991.)

Q#7 (15%). Include the snapshot accuracy assessment report

7 Unsupervised Classification

Lab developed by Rong Fang from Dr. Strimbu and ENVI documentation.

7.1 Objectives

Generate classification image by using supervised and unsupervised classification and assess the accuracy with confusion matrix

7.2 Files used in this lab

Files (.dat and .hdr format)	**Description**
PostFireOLISubset	Landsat 8 OLI bands (seven total), from May 25, 2014

The images are related with the wildlife that occurred on May 2014 in San Diego County, CA. In May of 2014, close to 20 different wildfires erupted in San Diego County, triggered by SantaAna winds and a heatwave. The first fire started on May 5, and the last remaining fires were extinguished by May 22. By May 18, the fires had burned more than 27,000 acres (42 square miles) of land (Figueroa and Winkley, 2014). Some of the communities affected by the fire included Camp Pendleton, Carlsbad, San Marcos, and Escondido.

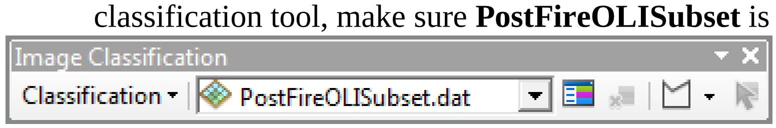
7.3 Processing

7.3.1 Supervised classification

Execute a maximum likelihood supervised classification of the Landsat 8 OLI bands image **PostFireOLISubset**. The classification should create 6 classes: **Water, Bare Land/ Cloud, Developed Area, Dense Vegetation, Sparse Vegetation, and Shrubs.** Assess the accuracy of the classification using the same procedure as you have done in the previous lab.

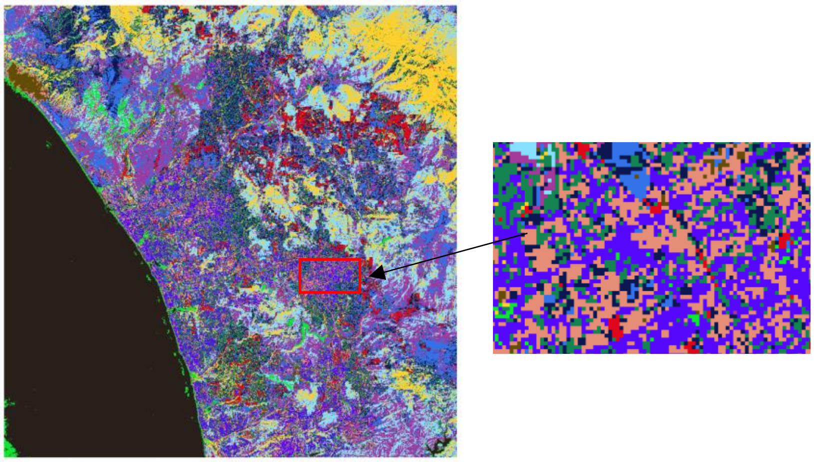
7.3.2 Unsupervised classification

1. Start ArcMap, add image ***PostFireOLISubset*** by using **Add Data** icon . Set the display of **PostFireOLISubset** as false-color (Red: Near-Infrared, Green: Red, and Blue: Green) for better visual identification.
2. Add the base map as a reference. Click **Add Data** icon , choose **Add Basemap →Imagery.**
3. Enable **image classification** tool and the **Spatial Analyst** is also active. In the image classification tool, make sure **PostFireOLISubset** is chosen as the image to be classified.



4. Click classification, choose **ISO Cluster Unsupervised Classification**. Start with **Number of classes 12.** Name the output classified raster as **PostFire12.**

The classification of 12 classes is too patchy and noisy. We can see that many patches are too small. In the Table of Content, right-click on the layer ***PostFire12***, and open the **attribute table.** Exam the classes with smallest count.



Q1 [10%]. How many counts are there in the smallest two classes?
Q2 [10%]. What land cover do you think is located in the classes with the smallest number of pixels?
Q3 [10%]. Is there any visually identifiable misclassification in these classes?

Now, please visually determine the majority of land cover type of these classes in the *PostFire12* file by comparing the classified image with the Base map.

Class Value	Land Cover Type
1	Water
2	Bare Land
3	Developed Area
4	Dense Vegetation
5	Sparse Vegetation
6	Shrub
7	Bare land
8	Developed Area
9	Developed Area
10	Shrub
11	Bare Land
12	Cloud/Bare Land

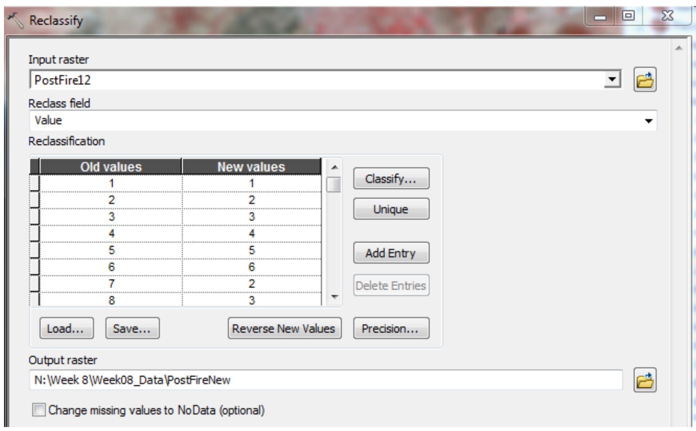
7.3.3 Reclass image

According to your judgment, do you think you need to merge some of the classes? In my case, I decided to merge class#2 7, and 11 as a single class Bare Land, 6, and 10 as Shrub, 3, 8 and 9 as Developed.

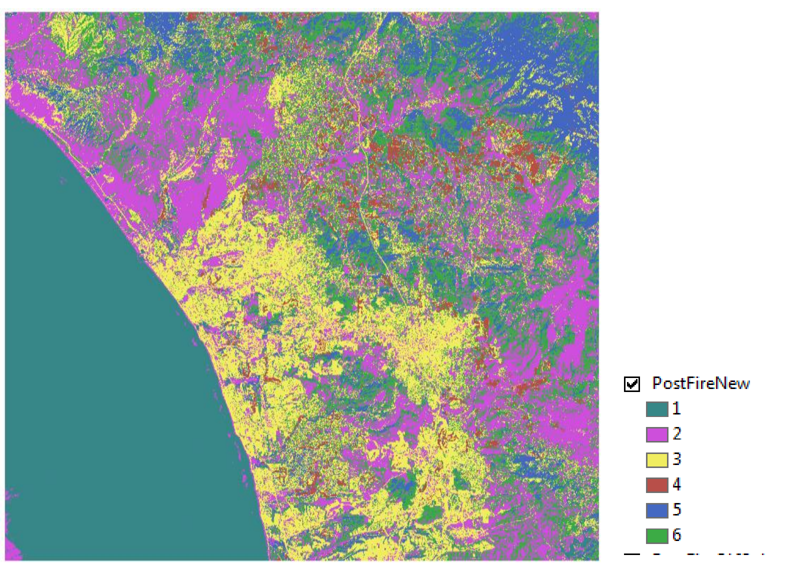
Final Classes	Original Classes	Land Cover Type
1	1	Water
2	2, 7, 11, 12	Bare Land/ Cloud

3	3, 8, 9	Developed Area
4	4	Dense Vegetation
5	5	Sparse Vegetation
6	6, 10	Shrub

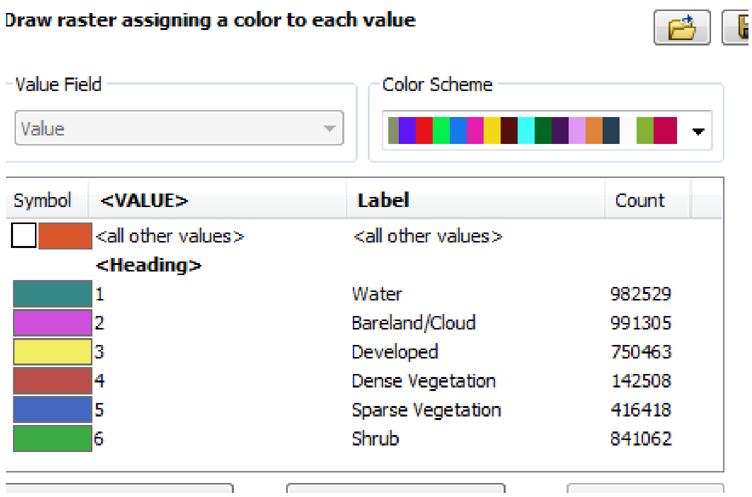
5. Open ArcToolBox, find Spatial Analyst Tools, and choose **Reclass →Reclassify**. In the dialog of Reclassify, input the parameters as below:
   Input raster: PostFire12
   Reclass field: Value
   Reclassification: Input the values according to your reclassification table.
   Output raster: PostFireNew



6. Click OK once you have finished entering the paprametrs. The reclassified image will be displayed.



7. Next step is to set the labels of the different classes to the land cover type. Right, Click on the layer **PostFireNew** in **Table of Content**. Choose **Layer Properties**. In the symbology tab, change the label to the land cover type.

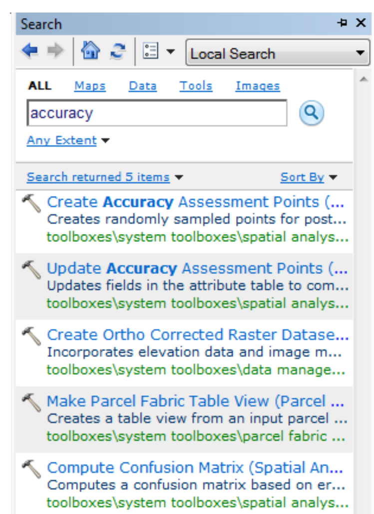


7.3.4 Accuracy assessment

7.3.4.1 Create Ground Truth Samples

You should use the same AOI that was created during the supervised classification.

7.3.4.2 Create Accuracy Assessment Points



In this step, we need to randomly sample some points on the Ground Truth polygons for the purpose of assessing the accuracy of the classification.

1. In the main menu, click to open the **search window**. Type **accuracy** in the **Search window**. Open **Create Accuracy Assessment Points**. Enter the parameters as below:

   Input raster or feature class data: PostFireNew
   Output Accuracy Assessment Points: GPoints
   Target Field: Ground_Truth
   Number of Random Points: 500
   Sampling Strategy: Equalized_Stratified_Random.

Click OK.

Table

GPoints

FID	Shape *	Classified	GrndTruth
0	Point	-1	5
1	Point	-1	4
2	Point	-1	3
3	Point	-1	6
4	Point	-1	3
5	Point	-1	4
6	Point	-1	2
7	Point	-1	5
8	Point	-1	4
9	Point	-1	3
10	Point	-1	5
11	Point	-1	2
12	Point	-1	3
13	Point	-1	2
14	Point	-1	6
15	Point	-1	2
16	Point	-1	5

2. After the points are generated, you will see these points covering on the top of ground truth polygons. Right-click the **GPoints** in the **Table of Content**, and open the **Attribute Table.** You will see -1 as the default value showing in the column of **Classified**.

3. Next step, we need to update accuracy assessment points and identify their corresponding value in the classified image. Type **Accuracy** in the Search window again. Open **Update Accuracy Assessment Points.** Enter the parameters as below:
   Input raster or feature class data: PostFireNew
   Input Accuracy Assessment Points: GPoints
   Output Accuracy Assessment Points: AccuracyAssement.shp
   Target Field: Classified

Click OK.

4. Once the points are updated, in the **Table of Content**, right-click the **AccuracyAssement** layer, open **Attribute Table**. You will see both column Classified and **GrndTruth** contain their values

Table

AccuracyAssement

FID	Shape *	Classified	GrndTruth
0	Point	1	1
1	Point	1	1
2	Point	2	1
3	Point	2	1
4	Point	2	1
5	Point	2	1
6	Point	2	1
7	Point	2	1
8	Point	2	2
9	Point	2	2
10	Point	2	2
11	Point	2	2
12	Point	2	2
13	Point	6	2
14	Point	2	2

7.3.4.3 Create Confusion Matrix

1. In the search Window, type **confusion matrix**. Open **Compute Confusion Matrix**.
   Input Accuracy Assessment Points: AccuracyAssessment;
   Output Confusion Matrix: UnClassi. Then click OK.

2. A .csv table is generated and added in the Table of Contents. Open the table UnClassi by right-clicking the **UnClassi**.scv file in the **Table of Contents**., you will see the table shows as below:

Table

UnClassi

OBJECTID *	ClassValue	C_1	C_2	C_3	C_4	C_5	C_6	Total	U_Accuracy	Kappa
1	C_1	106	0	0	0	0	0	106	1	0
2	C_2	295	41	13	0	0	0	349	0.117479	0
3	C_3	0	0	12	0	0	0	12	1	0
4	C_4	0	0	0	9	0	0	9	1	0
5	C_5	0	0	0	1	9	4	14	0.642857	0
6	C_6	0	9	3	0	1	6	19	0.315789	0
7	Total	401	50	28	10	10	10	509	0	0
8	P_Accuracy	0.264339	0.82	0.428571	0.9	0.9	0.6	0	0.359528	0
9	Kappa	0	0	0	0	0	0	0	0	0.163508

The column ClassValue is the value in the classified image, and different columns of c_1 to c_6 are the ground truth value. User's accuracy shows false positives, where pixels are incorrectly classified as a known class when they should have been classified as something else. User's accuracy is also referred to as errors of commission, or type 1 error. Producer's accuracy is a false negative, where pixels of a known class are classified as something other than that class. Producer's accuracy is also referred to as errors of omission, or type 2 error. Kappa index of agreement gives an overall assessment of the accuracy of the classification

Q4 [10%]. Briefly describe how does ISO-DATA algorithm determines the classes?
Q5 [10%]. For supervised classification, what is the class with smallest and largest user accuracy?
Q6 [10%]. For unsupervised classification, what is the class with smallest and largest user accuracy?
Q7 [10%]. For supervised classification, what is the class with the least producer accuracy?
Q8 [10%]. For unsupervised classification, what is the class with the least producer accuracy?
Q9 [10%]. What does cause the low producer and user accuracy?
Q10 [10%]. What is the overall accuracy and Kappa statistics for the two classifications: supervised and unsupervised?

7.4 Advanced topics: Unsupervised classification and confusion matrix with ERDAS

Lab developed by Sudeera Wickramarathna and Dr. Strimbu from ERDAS documentation

7.4.1 Objectives

- Generate classification image by using suppervised and unsupervised classification.
- Assess the accuracy with confusion matrix

7.4.2 Files used in this Lab

The file needed to complete this Lab are supplied by the U.S. Geological Survey, and are located on Canvas, Week 8 Lab. Copy the files to a local drive, such as the Desktop.

Files (.dat and .hdr format)	**Description**
PostFireOLISubset	Landsat 8 OLI bands (seven total), from May 25, 2014

The images are related to the wildlife that occurred in May 2014 in San Diego County, CA. The first fire started on May 5, and the last remaining fires were extinguished by May 22. By May 18, the fires had burned more than 27,000 acres (42 square miles) of land (Figueroa and Winkley, 2014).

7.4.3 Supervised classification

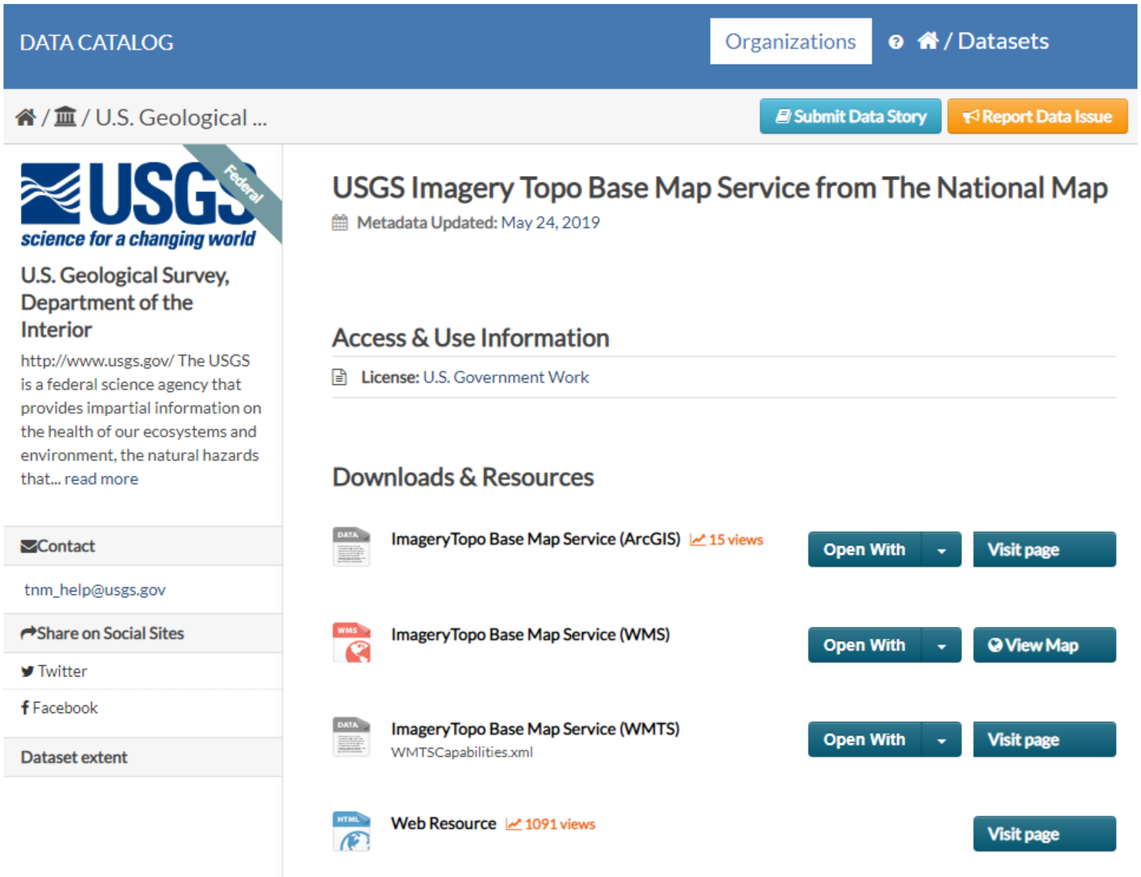
Using the knowledge acquired in the previous Lab you should classify the image using the supervised classification method. You should consider the following six (6) classes:

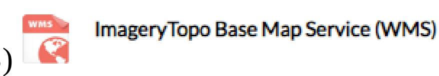
Class Value	Land Cover Type
1	Water
2	Bare Land/ Clouds
3	Developed Area
4	Dense Vegetation
5	Sparse Vegetation
6	Shrub

Because the new area to be classified is located inside USA a basemap service is available within ERDAS. The service can be accessed thru the Web Map Service (WMS) available in ERDAS. To connect the WMS to ERDAS you have to provide ERADS with the location of the server. One such location is https://basemap.nationalmap.gv/arcgis/services/USGSImageryOnly/MapServer/WMSServer?request=GetCapabilities&service=WMS.

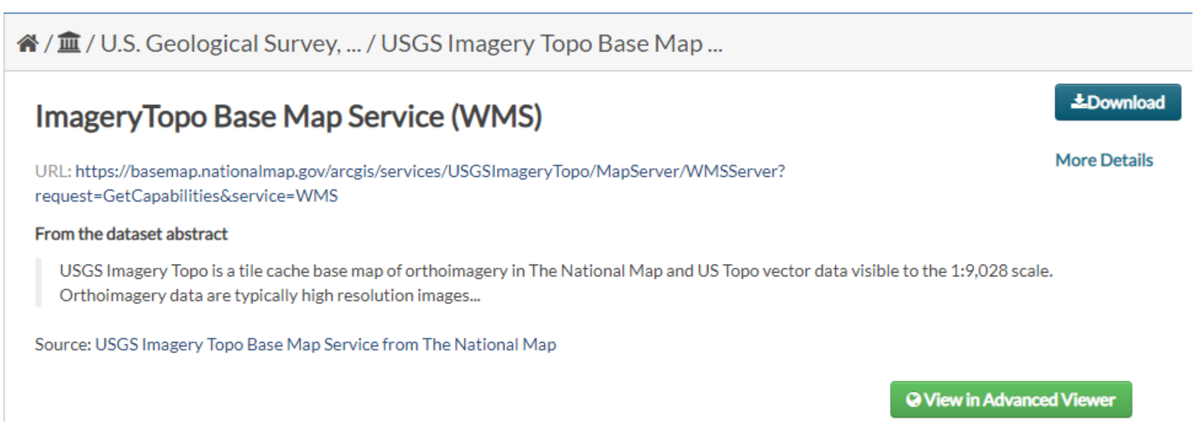
Another link hosting the same information can be accessed by searching with Google “basemaps usgs server” which would supply a series of results. The imagery can be accessed from the viewer.nationalmap.gov or from the catalog.data.gov.

If you want to use the catalog.data.gov website, once you click on it, an interface like this could appear (depends on where you are pointed inside the website):

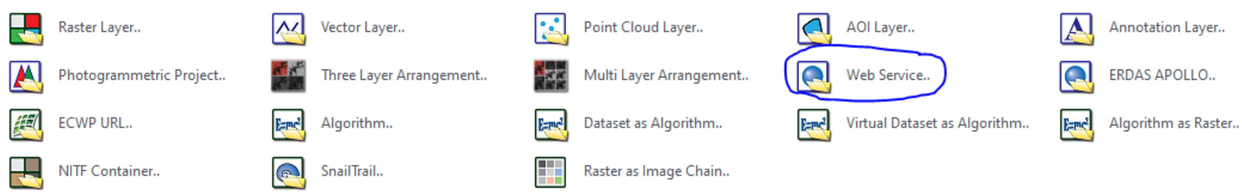


You should click the Imagery Topo Base Map Service (WMS) 

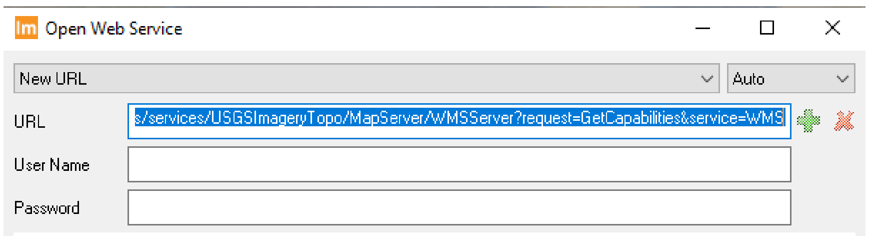
, which will open a new page that contains the address of the server:



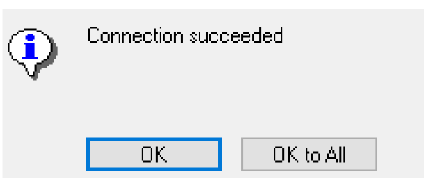
Copy the url, then open ERDAS. Select File->Open, and from the available option choose Web Service…



Within the new window paste the address of the server:

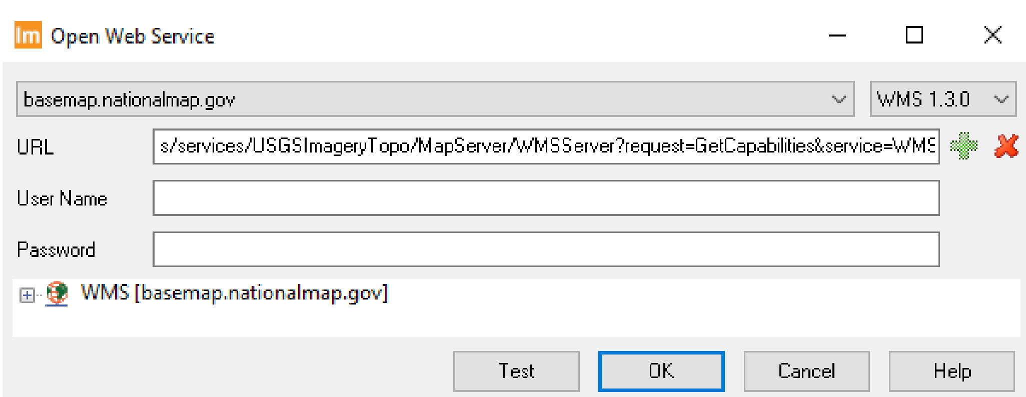


To check the connection with the server, press Test. If the connection was successful a new window should appear stating this:

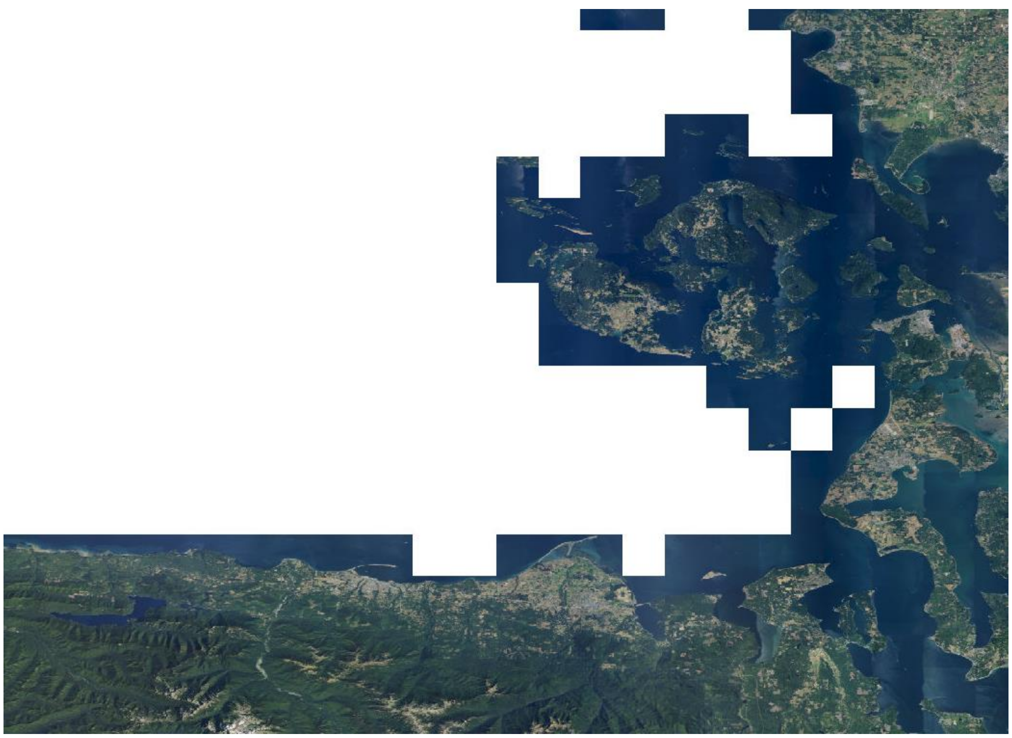


If the connection was successful, the WMS link will be display below the Password. To permanentize the connection, such that will be used later, you should press the green cross, located at the right of the url box .

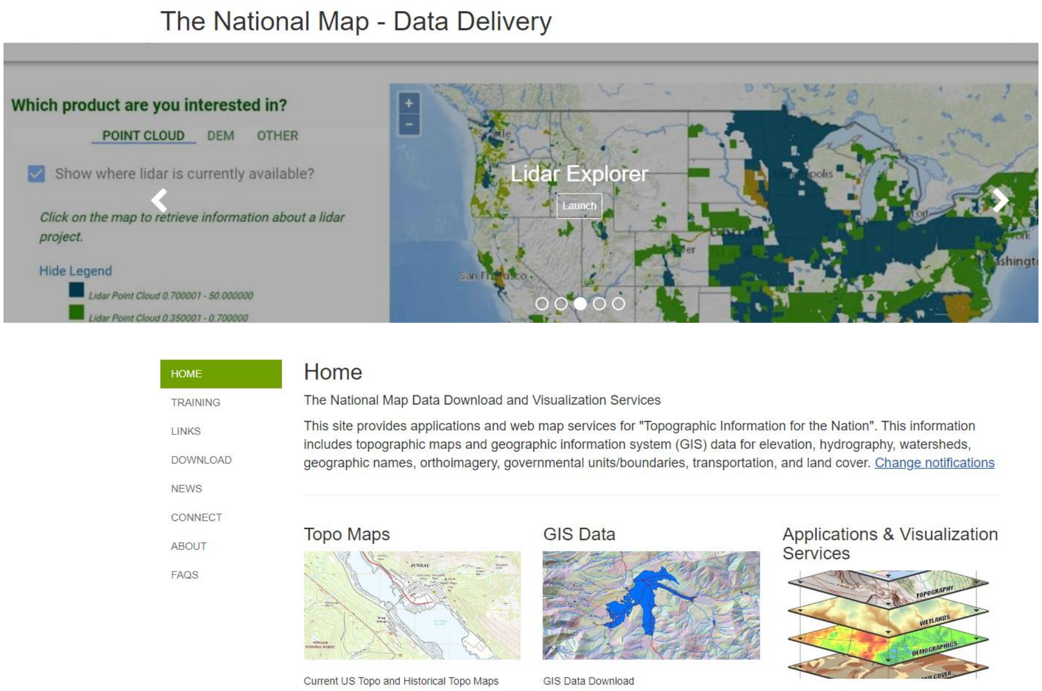
After the cross is pressed, the name of the server will change from New URL to the WMS name:



Once the connection was added to the WMS list, you can click OK. You will see the entire planet, but detailed imagery is available only for the USA. For example, the last lab, used a Landsat image that covered parts of USA and parts of Canada. If you zoom in to the Vancouver Island- Olympic Peninsula area, you will notice that only the USA is rendered:



If you select the viewer.nationalmap.gov then you will be directed to the National Map data delivery website:



From the website you should choose the Apps & Services link:

In the new webpage, you should choose Services, then Web Map Services.

Among the available services, you should choose Base Maps, then select USGS Imagery Topo Base Map, the WMS link:

Base Maps (Cached)					
Base Map - Blank					
REST	WMS	ArcGIS.com	ArcMap	Legend	Thumbnail
Hydrography - Tile Cache					
REST	WMS WMTS	ArcGIS.com	ArcMap	Legend	Thumbnail
USGS ImageryOnly Base Map - Primary Tile Cache					
REST	WMS WMTS	ArcGIS.com	ArcMap	Legend	Thumbnail
USGS Imagery Topo Base Map - Primary Tile Cache					
REST	WMS WMTS	ArcGIS.com	ArcMap	Legend	Thumbnail
USGS Shaded Relief - Primary Tile Cache					
REST	WMS WMTS	ArcGIS.com	ArcMap	Legend	Thumbnail

Clicking WMS link will open an html interface, from which you should select only the link to the server: https://basemap.nationalmap.gov:443/arcgis/services/USGSImageryTopo/MapServer/WmsServer?

From the code written there:


```
<WMS_Capabilities xmlns="http://www.opengis.net/wms" xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance" xmlns:esri_wms="http://www.esri.com/wms" version="1.3.0" xsi:schemaLocation="http://www.opengis.net/wms
http://schemas.opengis.net/wms/1.3.0/capabilities_1_3_0.xsd http://www.esri.com/wms https://basemap.nationalmap.gov:443/arcgis/services/USGSImageryTopo/MapServer/WmsServer?
version=1.3.0%26service=WMS%26request=GetSchemaExtension">
 <Service>
  <Name>
   <![CDATA[ WMS ]]>
  </Name>
  <Title>WMS</Title>
  <Abstract>WMS</Abstract>
  <KeywordList>
   <Keyword>
    <![CDATA[ ]]>
   </Keyword>
  </KeywordList>
  <OnlineResource xmlns:xlink="http://www.w3.org/1999/xlink" xlink:type="simple" xlink:href="https://basemap.nationalmap.gov:443/arcgis/services/USGSImageryTopo/MapServer/WmsServer?"/>
  <ContactInformation>
   <ContactPersonPrimary>
    <ContactPerson>
     <![CDATA[ ]]>
    </ContactPerson>
    <ContactOrganization>
     <![CDATA[ ]]>
    </ContactOrganization>
   </ContactPersonPrimary>
   <ContactPosition>
    <![CDATA[ ]]>
   </ContactPosition>
   <ContactAddress>
    <AddressType>
     <![CDATA[ ]]>
    </AddressType>
    <Address>
     <![CDATA[ ]]>
    </Address>
    <City>
     <![CDATA[ ]]>
    </City>
    <StateOrProvince>
     <![CDATA[ ]]>
    </StateOrProvince>
    <PostCode>
     <![CDATA[ ]]>
    </PostCode>
    <Country>
     <![CDATA[ ]]>
    </Country>
   </ContactAddress>
   <ContactVoiceTelephone>
    <![CDATA[ ]]>
```


After you copy the link to the WMS, paste it in the interface of adding Web Map services in ERDAS, and follow the similar steps as you did before.

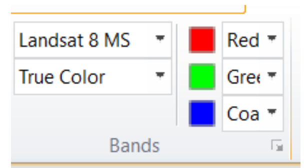
Q1 (30%): Include a series of snapshots with the main steps used for supervised classification.

7.4.4 Unsupervised classification

It is customary to classify images using supervised classification and unsupervised to acquire a sense of the best spectral separation of the pixels within the image. Therefore, in this Lab we will implement an unsupervised classification using the ISODATA classification method, developed by Ball and Hall in 1965. The ISODATA algorithm, an abbreviation of Iterative Self-Organizing Data Analysis Technique, contains a series of operations that merge pixels or clusters of pixels if their separation distance multispectral feature space is less than a predefined value. The ISODATA algorithm also split a cluster into two clusters according to user-preset rules.

1. Download the image files and locate them in your Lab 7 data folder on your computer. **Next, Click File > Open > Raster Layer** and select **"PostFireOLIsubset.hdr".** Now you can see the newly added image in the Content pane and **2D View#1** as well.

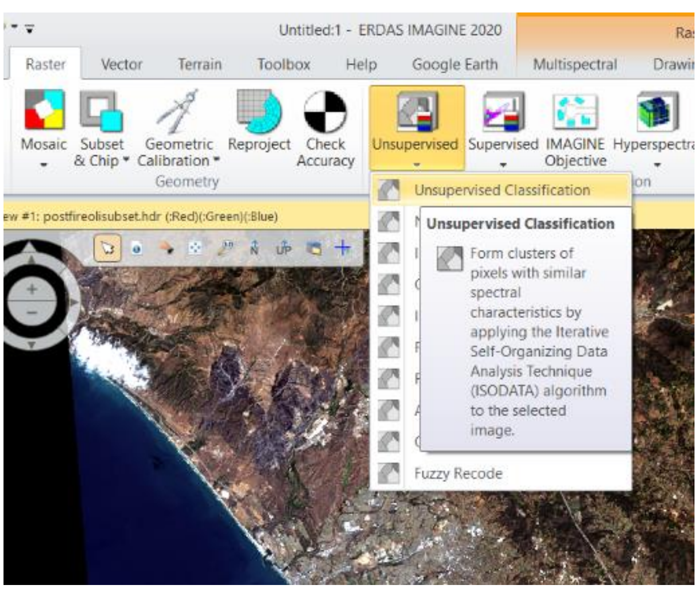
2. Next, **Click Multispectral** Tab and go to **Bands group.** You will see **Landsat 8 MS** in the Sensor Type window; if not, change it to Landsat 8 MS by clicking the drop-down menu option. Also, change the **Band Combination as True Color.**

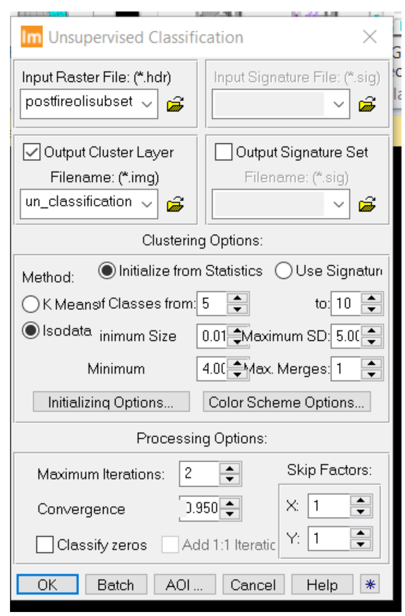


3. To access the **ISODATA algorithm**, select **Unsupervised under the Raster Tab.** Click **Unsupervised Classification** from the drop-down menu. Now you will see the **Unsupervised Classification dialog box.**

4. Click the **Output Signature Set** Checkbox and give the **Output Cluster Layer** and the **Output Signature Set** a similar name (i.e., **Unsupervised_classification**).

5. Also, make sure that under **Clustering Options,** the **Initialize from Statistics** box is **selected.**





6. The Unsupervised Classification window asks for the parameters defining the ISODATA algorithm, specifically the number of classes and the variability within a class. However, there are several additional options that you should specify. To mirror the unsupervised classification**, you should choose the number of classes to range from 5 to 10, as subsequent merging could be needed.**

7. Here, the **Number of Classes from** shows the lower limit for the number of classes to be created while the **Number of Classes to** show the upper limit for the number of classes to be created.

8. Because ISODATA is an iterative algorithm, you should choose the number of iterations. For this Lab, you should **select 2**. The iterative process ends when the stopping rule is achieved.
   NOTE: Maximum Iterations is the number of times that the program will re-cluster the data. By setting up an appropriate number of iterations, you can prevent the utility from running too long, and also it will stop getting stuck in a cycle.

9. ERDAS uses a "Convergence" threshold to end the iterative process, which stops the iterations when the number of pixels in each class changes by less than the threshold. The ISODATA classification ends when either this threshold is met,or the maximum number of iterations is reached. This threshold prevents the Isodata utility from running indefinitely (Source: ERDAS IMAGINE help)

It makes sense that classes with few pixels should not exist, as probably are an artifact rather than a reality. **Minimum Size** field provides you to decide the size requirement for a cluster to be deleted at the end of an iteration. The size is measured as a percent of pixels in a cluster relative to the total number of pixels in the image. If there are fewer than the minimum size (e.g., 2%) of pixels in a class, ERDAS will delete that class, and the pixels will be placed in the class(es) nearest to them spectrally. Choose 1% as the Minimize Size for a class to be deleted (Source: ERDAS IMAGINE help).

10. To ensure that spectrally similar clusters are included in any class, a homogeneity measure of each class should be specified. A maximum class standard deviation should be entered in the **Maximum SD** field. You can keep the default value **5.00** for this field. If a class's standard deviation is larger than this threshold, then the class is split into two classes.

To ensure that classes are not only split but also merged, if they have similar spectral signatures and variability, you should select a minimum distance between class means and the maximum number of merge pairs. If the distance between class means is less than the minimum value entered, then ERDAS merges the classes. The maximum number of class pairs to merge is set by the maximum number of **Merge Pairs parameter**.

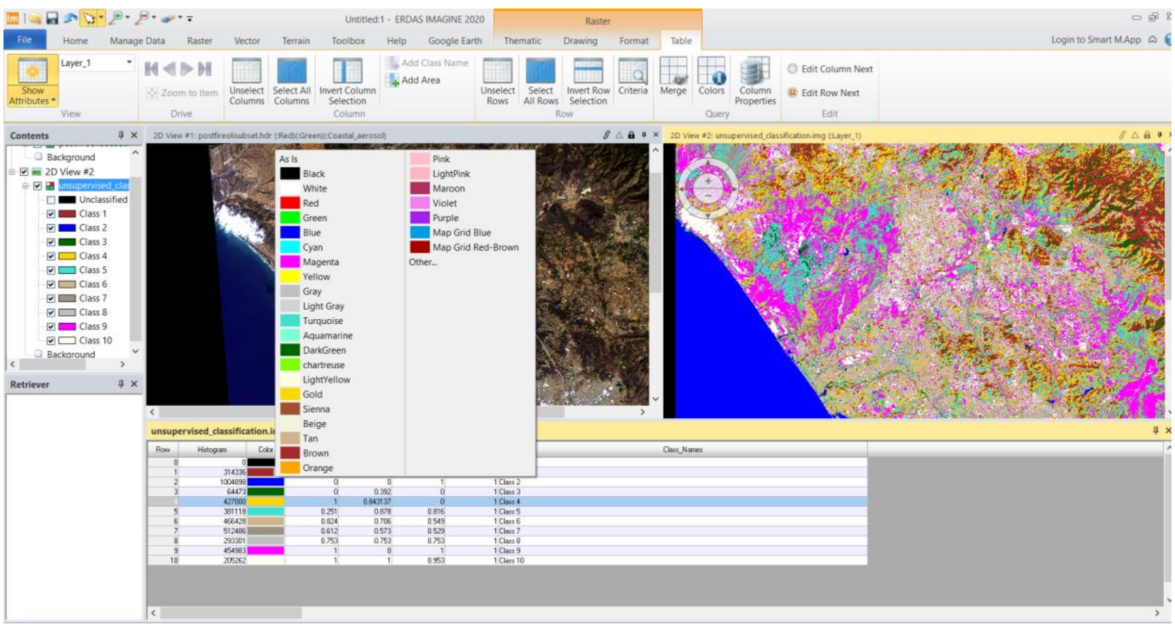
11. You could leave the **default values for the minimum class distance (i.e., 4) and the maximum Merges of merge Pairs (i.e., 1).** Click **OK.**

12. Now you can add a second View to display the classified image: Home > Add Views – Create New 2D View. Then left-click in the empty view window, select **Open Raster Layer** to add the classified image you just crested (**Unsupervised_classification.img)** by navigating to the data folder.

NOTE: You can determine the class number for each pixel by using the Inquire option in the information group under the **Home Tab**. A cursor display box will appear at the bottom of your main window. You can use the **pan tool** to select the pixel you are interested in.

13. Click on **Home** > **Link Views** icon drop-down > **Link Views.** Activate the **Equalize Scales** option as well. Pan the original image so that you can see the respective class of each pixel in the classified image (at the same geographic location in each image).

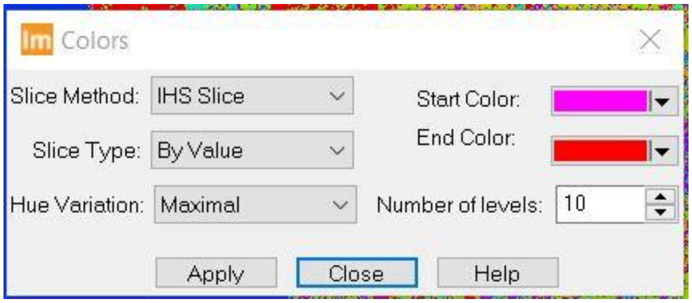
The following results could be obtained:



NOTE: You can change the color for each class by using the Attribute Table (at the bottom of your main window) color option.

To perform this task, you can go to the **Raster | Table** Tab and click the **Show Attribute** option. You will see an attribute table at the bottom of your main window. Left-click on the color tab for each class and select the appropriate color from the color pallet.

NOTE: Alternatively, you can try **table**> **Colors** in **Query group** to assign color ramp. You can select **Start Color and End Color**; ERDAS will create a color ramp for you.



Q2(15%): Include a snapshot of your classified image at this stage. Also, include another snapshot of the attribute table.

14. As you see, the number of classes produced is 10, which is larger than the desired number. Therefore, some merging should be done. But before any merger is executed, you should compare the result with the supervised image and decide if the classification meets your visual requirements. (you can synchronize the views).

Row	Histogram	Color	Red	e	lu	Opacity	
0	0		0	0	0	0	Unclassified
1	314336		0.647	:5	:5	1	Class 1
2	1004898		0	0	1	1	Class 2
3	64473		0	:2	0	1	Class 3
4	427000		1	:3	0	1	Class 4
5	381118		0.251	'8	6	1	Class 5
6	466428		0.824	l6	:9	1	Class 6
7	512486		0.612	'3	:9	1	Class 7
8	293301		0.753	:3	:3	1	Class 8
9	454983		1	0	1	1	Class 9
10	205262		1	1	:3	1	Class 10

15. If not, and most likely, this will be the case, rerun the classification by changing some parameters, such as the **Maximum size (%), Maximum SD, Minimum Distance, and Maximum Merges.**

A given class/cluster may represent more than one land cover class in the Ture color image. You can decide if there is a major or small shift between the classified image and the Ture color image. So, you should try several sets of parameters to ensure that the classified image is close to your expectations. Once you made your selection for the classified image, you should start improving it.

As you see, you have no unclassified pixels because you have chosen the option that forces ERDAS to classify all pixels. For example, examining the Attribute table, you noticed that all classes have more than 300,000 pixels, except for class 3, which has less than 70,000. Considering that a pixel's size is 30 m x 30 m, this class covers more than 15,000 ac. You could leave the class as it is.

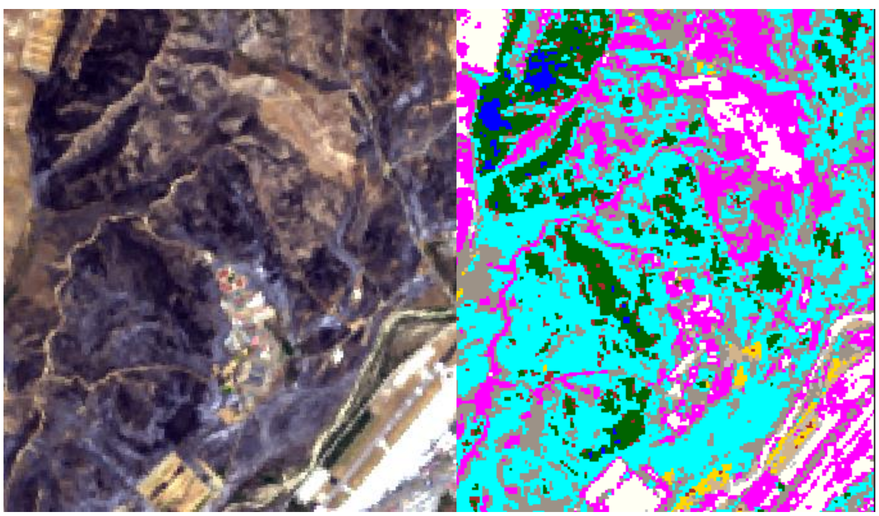
However, to mirror the supervised classification results, which has only 6 classes, you should merge some classes. To identify the classes to be merged, you should compare the classified image with the Landsat image and see similarities between them. To merge classes, you should focus on smaller classes, which are likely candidates to be merged. Therefore, you will start with class #3.

16. Now you can remove the secondary view (i.e., 2D View#2, classified image window). Go to **Contents pan, select and right-click on 2D View#1**. Then add the **Unsupervised Classified** file to it.
17. Click the **Home tab** and go to **View group,** and select **Swipe option** 

. Alternatively, you can go to **Utility | Transition Tab** and select different **Swipe options. You can also add the Swipe line** so that you can see the Transition between two images.

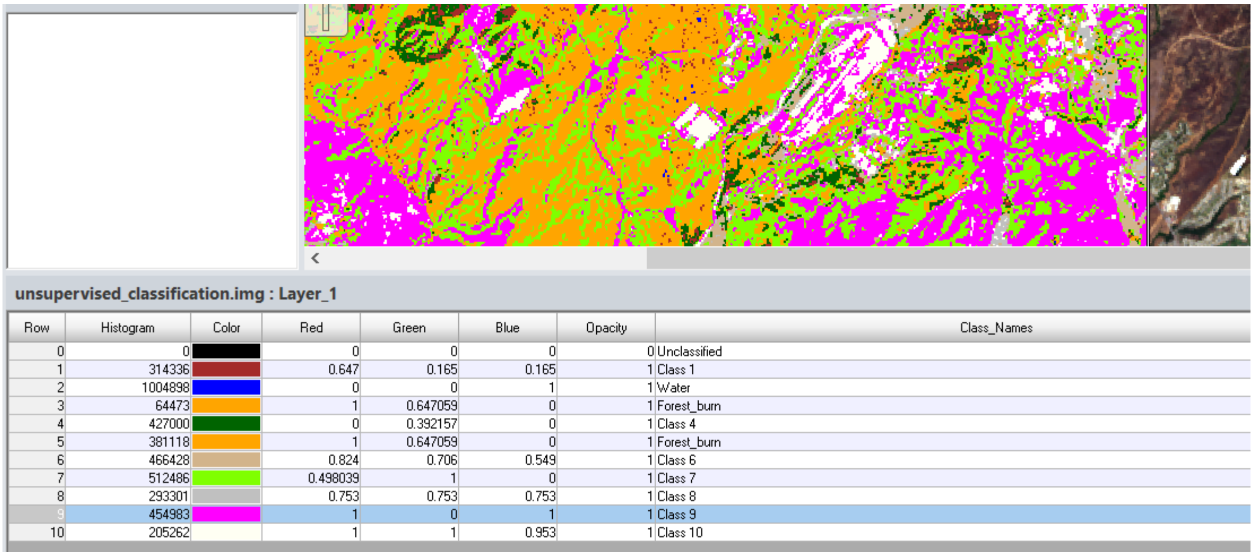
18. Now you will see the unsupervised classified image top of the True Color image (**"postfireolisubnet.hdr"**). You can swipe the top image and can observe the features of the True color image.

19. By swiping the images, you noticed that class#3 reflects the areas that suffered intense burning (Green colored patches). Therefore, if you are interested in burn severity, you could leave it as a separate class. However, if you are interested only in the burn vs. not burned, then you could merge it with class#5 (Cyan colored patches)



Class 3 and 5 reflect burns that happened.

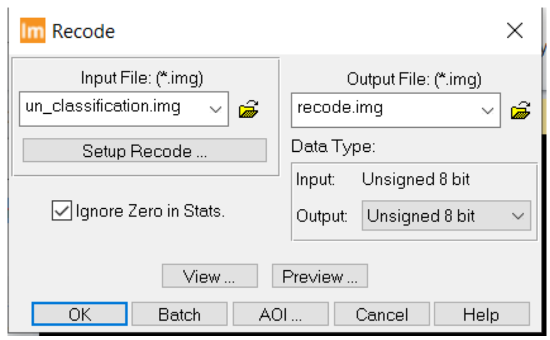
20. Likewise, you can find classes that can be combined as a single class. Once you have identified combined classes, rename them (e.g., Class 3 and 5 renamed as Forest_burn) in the **Attribute table** and assign a color as well as follows:



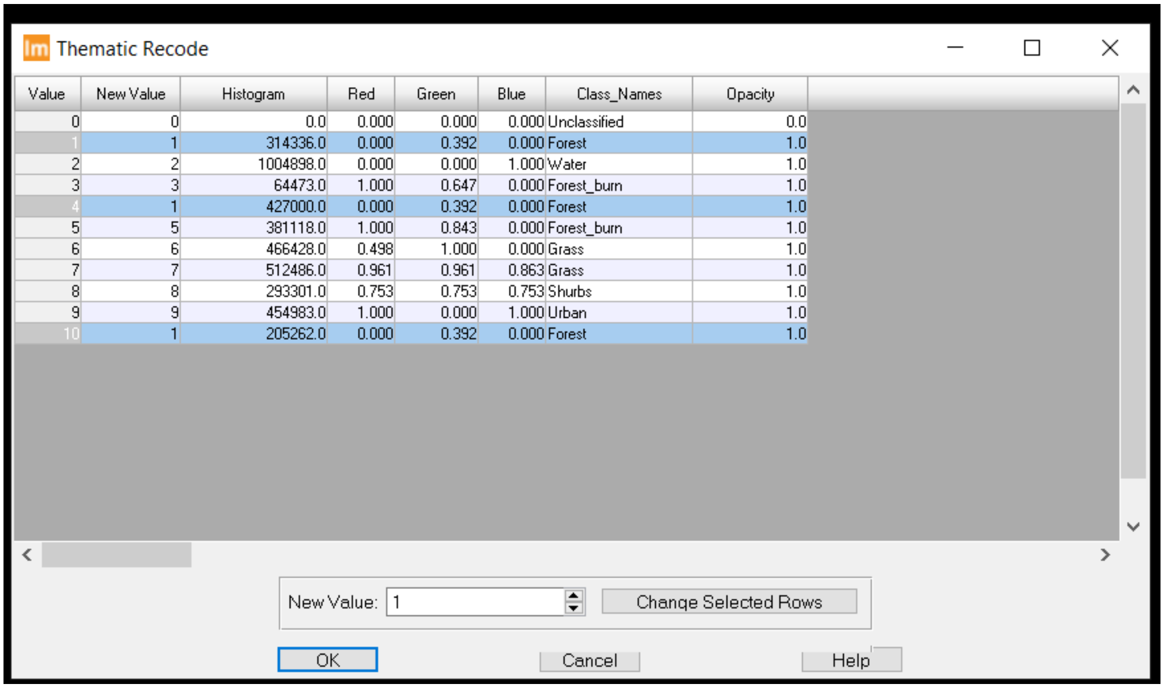
Row	Histogram	Color	Red	Green	Blue	Opacity	Class_Names
0	0		0	0	0	0	Unclassified
1	314336		0.647	0.165	0.165	1	Class 1
2	1004898		0	0	1	1	Water
3	64473		1	0.647059	0	1	Forest_burn
4	427000		0	0.392157	0	1	Class 4
5	381118		1	0.647059	0	1	Forest_burn
6	466428		0.824	0.706	0.549	1	Class 6
7	512486		0.498039	1	0	1	Class 7
8	293301		0.753	0.753	0.753	1	Class 8
9	454983		1	0	1	1	Class 9
10	205262		1	1	0.953	1	Class 10

21. Once you have finalized the classes by assigning the names and colors, **click save to save the file**. Then uncheck the true color image in the **Content pane**.

22. For example, to merge class#3 with class#5, you could use the **Thematic > Recode** option under the **Raster GIS** group located in **Raster Tab**. The recode dialog box will open. Navigate to the data folder and use **"unsupervised_classifcation.img"** as input file. Name output file as **"recode.img"** in the same data folder. Also, activate **Ignore Zero in Stats.** Option.

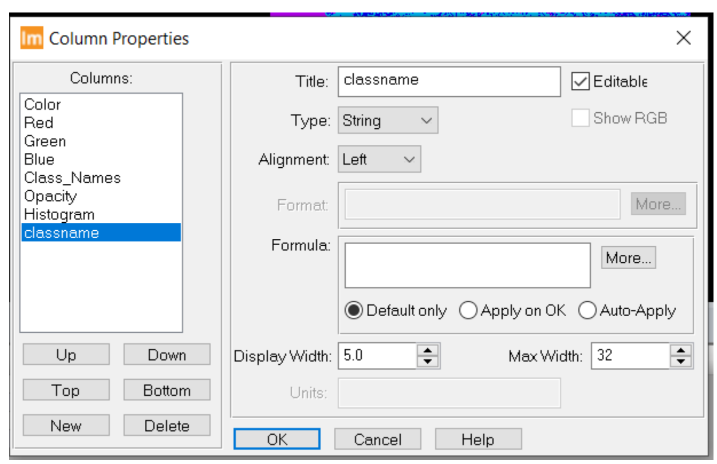


23. Then click **Setup Recode,** and another window will open with all the updates you just made to the **Attribute table.**
24. You can update the new table using following steps:
    1. Select a class (e.g., Forest)
    2. Select the row in this class by clicking them in the **Row column;** if you have more than one row you can use **Shift + Row Click** to select them all.
    3. Then change the value in the **New Value box**. You can use a desired unique value for this class (e.g., Forest = 1)
    4. Finally, select **Change Selected Rows**. This will change any row(s) that have been selected to a **New Value.** You can observe the changes in the **New Value column.**
    5. You can continue this process until you finalized your classes. Once you have finished updating the process, make sure there are no rows that have not been selected in the Recode Table. You can do this by right-clicking the **Value Column and select None.**
25. Click **OK** in **Thematic Recode table** and again click on in **Record Table** to crate your **Record file.**



Record File

1. Now, remove all images from the 2D Views that you have been working with so far.
2. Reload the **Record file** you just saved and open the **Attribute Table**.
3. You may notice this attribute table does not have a **Class_Name attribute column**. Now you can add a new attribute column by using the **table> Column properties in the Query Group.**
4. You can click **New** and type an appropriate title (e.g. "Class_Name") and select **String as Type. Click OK.**
5. A new column will appear, and you can name the classes with appropriate names. Also, you can change the class colors as well.
6. Now Save your changes by clicking the **Save** button.



Q#3 (15%): Include a series of snapshots with the main steps used in unsupervised classification

Q#4 (10%): Is unsupervised classification an iterative process or not? Justify your answer.

Q#5 (10%): Which classification is more accurate: supervised or unsupervised? Justify your answer.

7.4.5 Classification accuracy

ERDAS has a built-in function that computes the classification accuracy, namely the confusion matrix.

In ERDAS you can compute the confusion matrix using the Signature file for ground truth. To execute this task, first, you have to create the signature file, as you did for the supervised classification. The new signature file created on the LandsatTM image, must contain the same classes as the classified image. If different classes are made, you have to manually match them with the classes from the image.
The confusion matrix is computed after you select the Confusion Matrix Using Ground Truth signature file. If there are unmatched classes between signature file and image classes, you should establish their correspondence manually. The output will be the entire confusion matrix with all the parameters computed for you:

You can refer to Lab6 Image Classification > Accuracy Assessment steps to perform an accuracy assessment task.

Q#6 (10%). What is the difference between omission errors and commission errors? What are the other names for these errors?

Q#7 (10%). Cut and paste the confusion matrix in the Word document that you would submit on Canvas. Compute the percentage of false positives and false negatives using the Confusion matrix. Based on the omission and commission errors, what is your conclusion regarding the accuracy of your classification?

8 Classification enhancement using Remote Sensing Indices

Lab developed by Dr. Strimbu.

8.1 Objectives

The week 8 lab has two objectives:

- classify an image using recorded spectral reflectance and derived indices.
- assess the accuracy of the classification using the confusion matrix

8.2 Files used in this lab

Files (.dat and .hdr format)	**Description**
Sentinel2bDetroitLake	Sentinel 2B image east of Salem from Oct 29, 2020

Q1 [5%]. What are the wavelengths of the Sentinel 2B instrument?
Q2 [5%]. What are the wavelengths included in the Sentinel2bDetroitLake image?
Q3 [5%]. What is the spatial resolution of the image?
Q4 [5%]. Do you think that a post processing of the image occurred? Justify your answer.

8.3 Classification using only spectral data

Execute a maximum likelihood supervised classification and an ISO-DATA unsupervised classification of the Sentinel 2B image Sentinel2bDetroitLake. Both classifications should have the same classes, which should contain at least the following classes: **Water, Bare ground/ Clearcut, Burned areas, Mature forest, Young Forest, and Clouds.**
Q5 [10%]. Include a snapshot of approximately 1 square mile for both, supervised and unsupervised.
Assess the accuracy of both classifications using a confusion matrix.
Q6 [20%]. Include the confusion matrices for both classifications.

8.4 Compute remote sensing indices

Image classification is enhanced when in addition to the spectral information other data is added to the classification. The easiest data to be added to the classification is the one based on the same information as the original image, namely the remote sensing indices. Remote sensing indices are combination of the reflectance in various bands. The most famous index is the normalized difference vegetation index (NDVI), which delineates the vegetation from other land features. The NDVI, which is the ration between a combination of near-infrared and red reflectance, is an important vegetation index because is coupled with some physiological parameters describing the vegetation, is species-specific, and reduces many forms of multiplicative noise (e.g., sun illumination differences, cloud shadows etc.).

$$NDVI = \frac{\rho_{NIR} - \rho_{Red}}{\rho_{NIR} + \rho_{Red}}$$

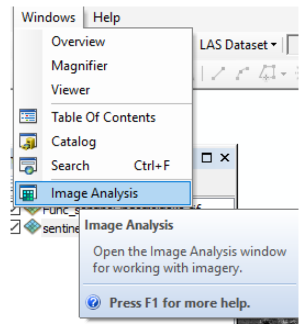
Beside NDVI, there are other indices that can be used to enhance image classification. In this lab we will be computing three more indices: transformed Modified Soil Adjusted Vegetation Index (MSAVI2), transformed Red-edge Vegetation Stress index (RVSI), and transformed normalized difference water index (NDWI):

$$MSAVI2 = \frac{1}{2}\left(2(NIR + 1) - \sqrt{(2NIR + 1)^2 - 8(NIR - Red)}\right)$$

$$RVSI = \frac{\rho_{704} + \rho_{782}}{2} - \rho_{740}$$

$$NDWI = \frac{\rho_{864} - \rho_{1610}}{\rho_{864} + \rho_{1610}}$$

The three indices are a transformation of the original indices, as the wavelengths do not match exactly the definition of the two indices.



1. To compute the two indices in ArcGIS you can use two options: Map Algebra and Image Analysis. The Map Algebra option is used when multiple rasters are combined, whereas the Image Analysis is used when the bands are merged into one image.

In this lab we will be using the Image Analysis tool, which combines the digital numbers from multiple bands within one image to produce a raster with only one band.

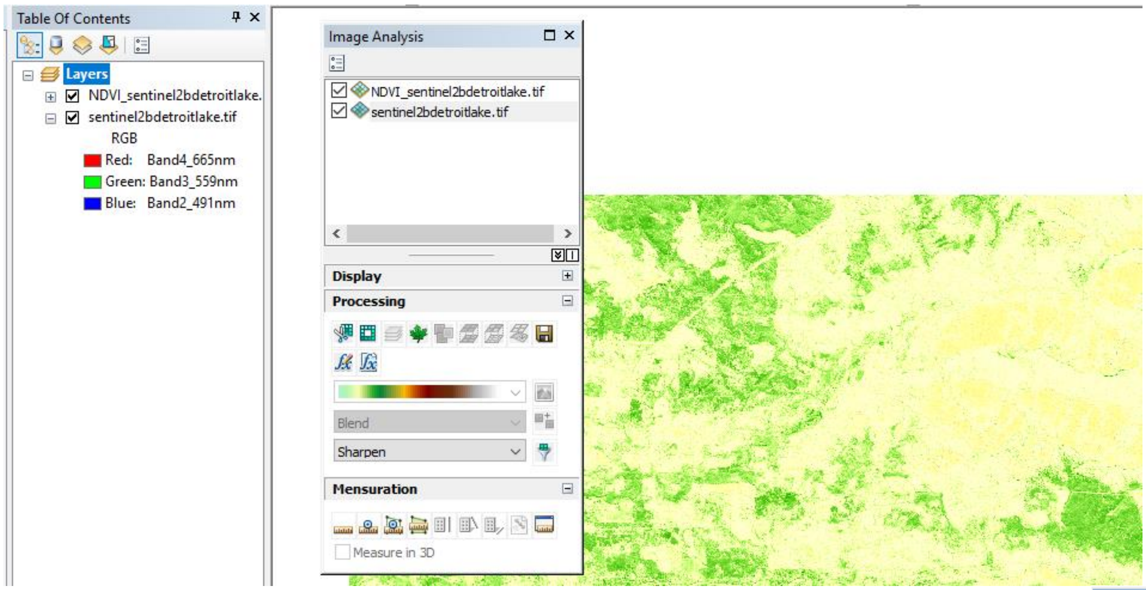
2. To turn on the Image Analysis, you simply select Windows from the menu bar followed by the Image Analysis option.

The image analysis window contains 4 areas: 1) the area where the rasters are located, 2) the Display area, which is used to enhance the displayed image, 3) the processing area, where various functions are developed, and 4) the Mensuration area, where measurements on the image can be done.

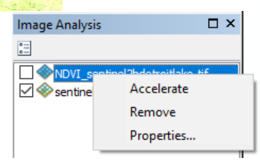
3. To compute various indices, you must combine the digital numbers of various bands into a new band. To achieve this task, first, you should choose the image subject to algebraic manipulations, which in this case would be the Sentinel2BDetrait Lake.
4. After you have selected the image, the Processing area will have many options turned on, one of which will be the NDVI button: . NDVI is not the only index readily available in ArcGIS, as many other are already implemented, however, the default option assumes a certain correspondence between the bands, such as the NIR would be band 4 and red band 3.

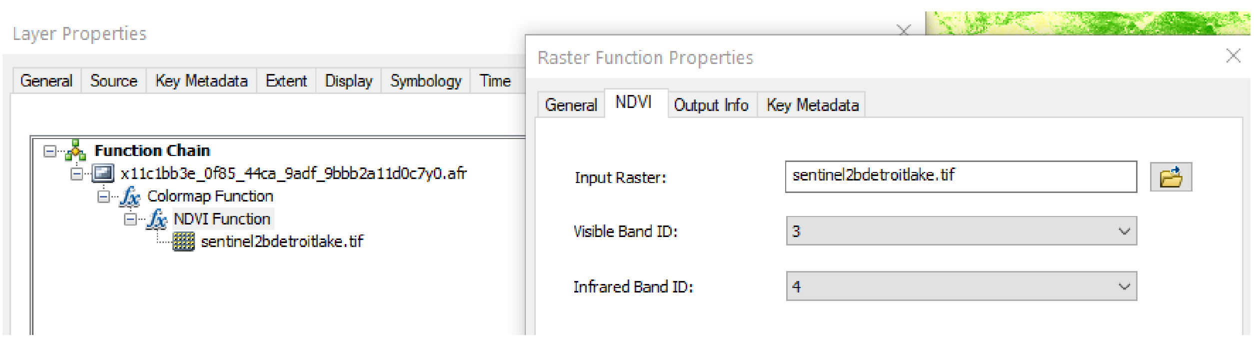
If the bands are not allocated according to the imbedded formula, the computations will be incorrect; therefore, always check the validity of the equation used to compute the index. In this lab we will be using the built-in NDVI index, and we will compute the NDWI by explicitly entering the formula.

The NDVI is immediately computed once the  is pressed, which will add the image to the Table of Contents and the raster to the image area:



5. To check the validity of the NDVI values, select NDVI_sentinel2bdetroitlake.tif from the Image Analysis window, then right-click when the mouse is over the name of the raster, and select the Properties … option. From the Layer properties window, select the Functions tab. Within the Functions window, right-click over the NDVI function, then choose the Properties… option.

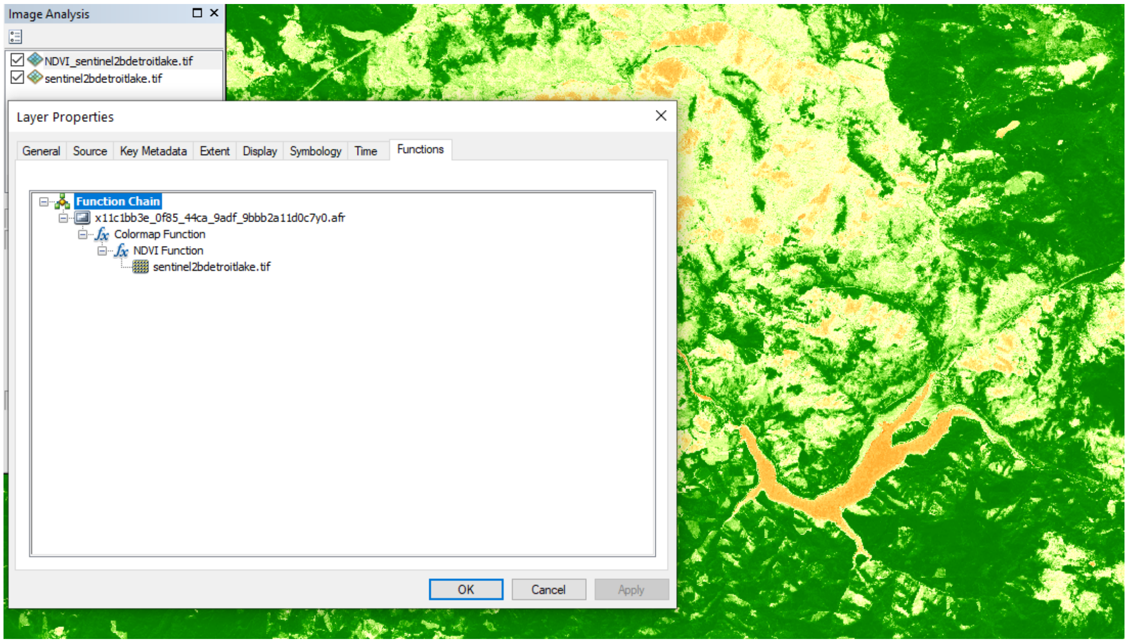




6. Because the NDVI formula is well established, the only check should be done if the NIR and red bands are properly identified within the list (stack) of available bands. The NDVI tab will show the bands that are used to compute the NDVI, which in this case is band 3 for Red and band 4 for Infrared.

7. Checking the specifications of the sentinel 2B instrument, you notice that band 3 is the red band from the Sentinel2bdetroitlake image, therefore it should be left unchanged. However, band 4 is not NIR, but the red-edge (i.e., 704 nm). In fact, the NIR, which formally starts at 750 nm wavelength, could be any of the band 6 to10 from the Sentinel2bdetroitlake image. It makes sense that among all the NIR bands to choose the one with the smallest wavelength, namely the 780 nm, which is band 6. Therefore, you should change band 4 from Infrared band to band 6, then press OK. To update the NDVI

computations and see them on the screen press Apply in the Layer Properties/ Functions tab.

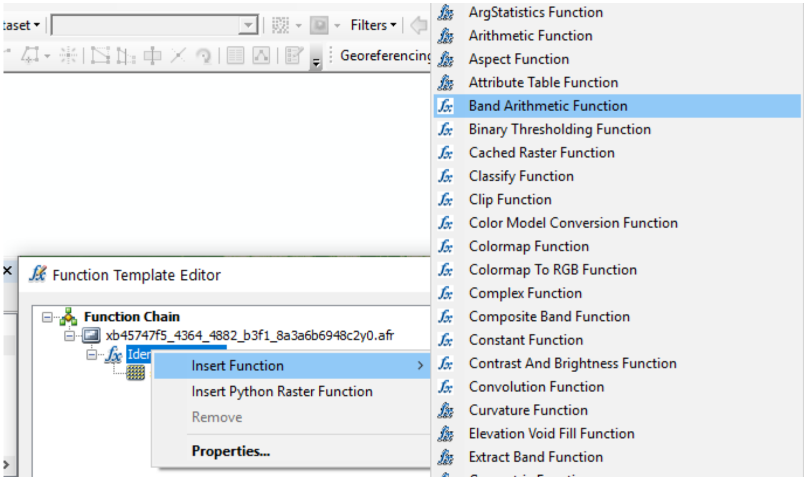


8. Once the image was updated, you can press OK in the Functions tab, as the NDVI computations are completed.

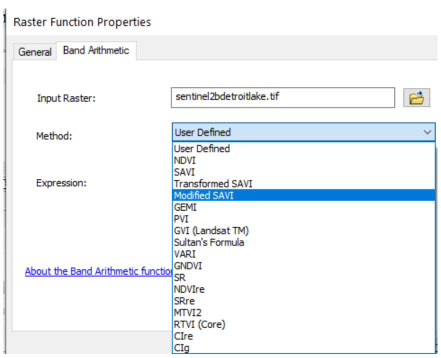
Q7 [5%]. Include a snapshot with the map showing the NDVI and the Functions tab from the Layer properties.

9. Next, let's compute the MSAVI2. The formula of the MSAVI2 is already implemented in ArcGIS, all that you have to do is access it. Therefore, again select the Sentinel2bdetroitlake image, but now instead of clicking the NDVI icon, you will select the function icon from the Processing area .

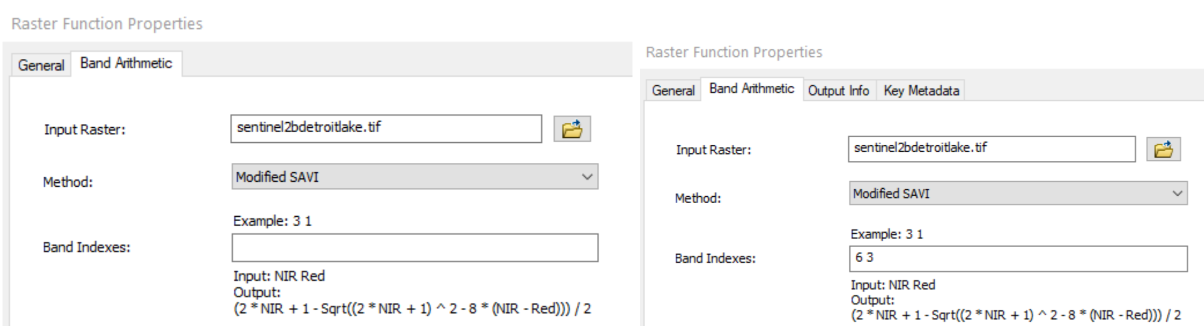
10. After you choose it, the default function is displayed in the Function Template Editor, which is normally the Identify function (will leave the original image unchanged). To access the MSAVI2 function, right-click on the Identify Function, then select Insert Function.



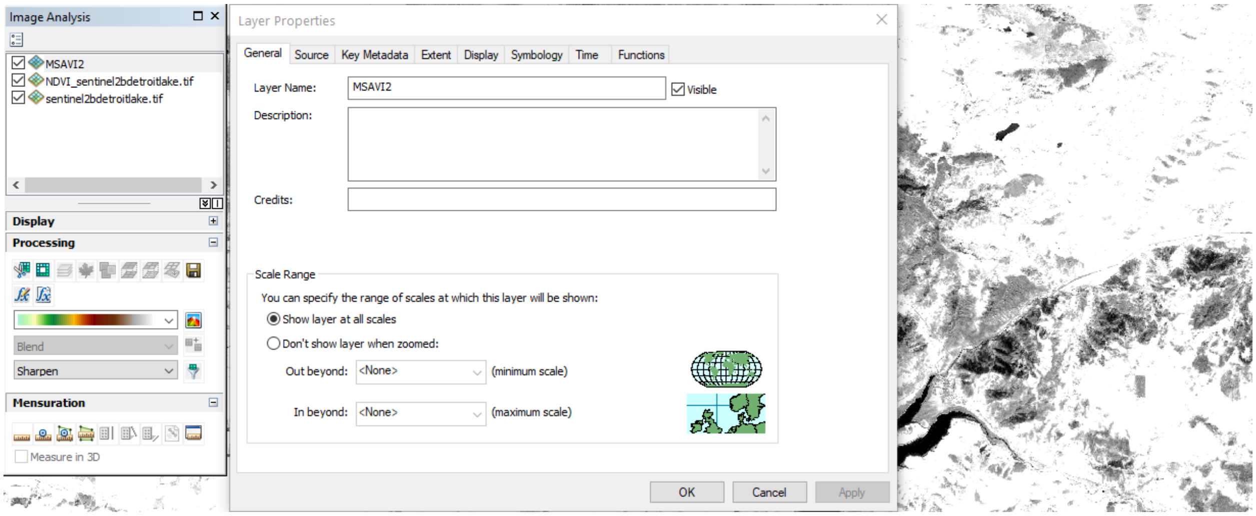
11. From the available functions, choose the Band Arithmetic Function. Within the Raster Function Properties, select the Band Arithmetic tab and make sure that the input raster is the Sentinel2bdetroitlake image, and choose from the dropped down Method, the MSAVI2, or Modified SAVI.



12. Once you have selected Modified SAVI, the formula and the corresponded bands are displayed. To properly allocate the NIR and Red bad, you should type "6 3", which means that band 6 is NIR, and band 3 is red.



13. After you press OK, a new image is added to the display, which shows the MSAVI2. If you want to change the name of the displayed layer, right-click on it in the Image Analysis window, then select Properties. Inside the General tab, change the Layer name to "MSAVI2", then click Apply.

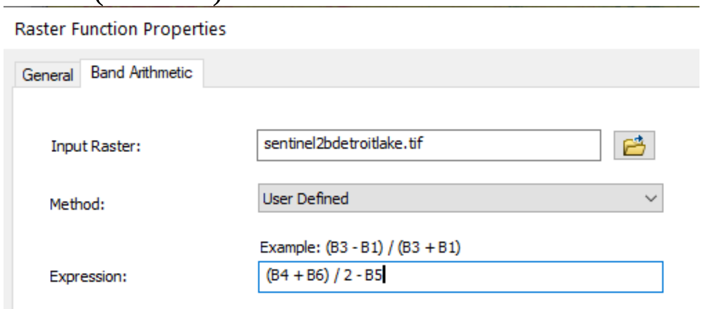


Q8[5%]. Include a snapshot with the Layer Properties showing the MASVI2 inside the Functions tab and the raster displaying the MSAVI2 using a color scheme different than the default grey ramp.

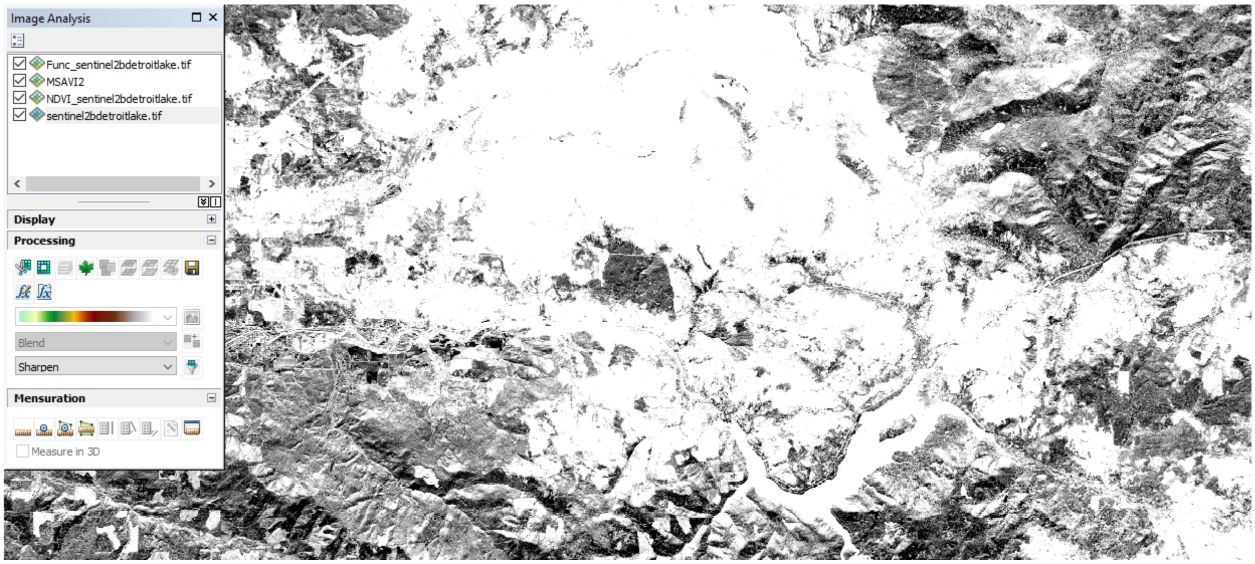
The last two indices to compute before the formal image classification starts are RVSI and NWSI. The two indices are computed in this case by entering the formula yourself using an approach similar to MSAVI2.

14. Therefore, first select the sentine2bdetraitlake image, followed by the function icon . Within the Function Template Editor right-click on the Identify Function, and from the functions available thru the Insert Function window also choose Band Arithmetic Function.

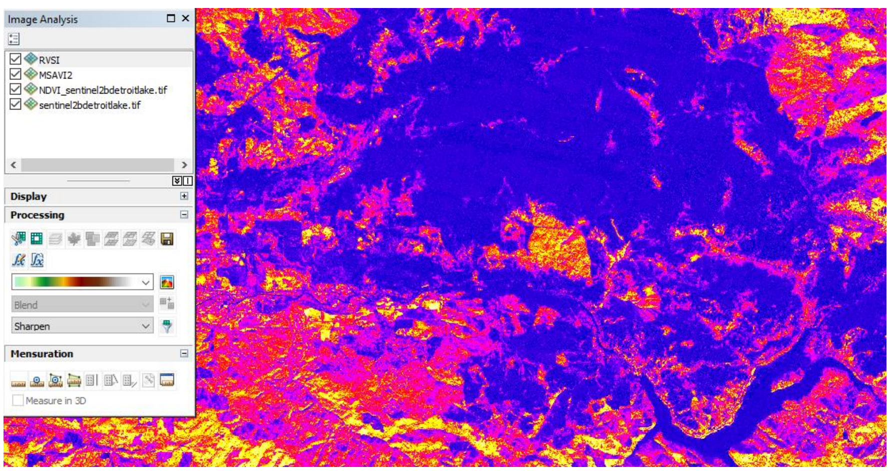
However, this time instead of searching for a predefined index, you should choose the User Defined option from the drop-down menu. In the expression box type in the formula for the RVSI: (B4 + B6) / 2 - B5



The image showing the RVSI values is displayed using a greyscale:

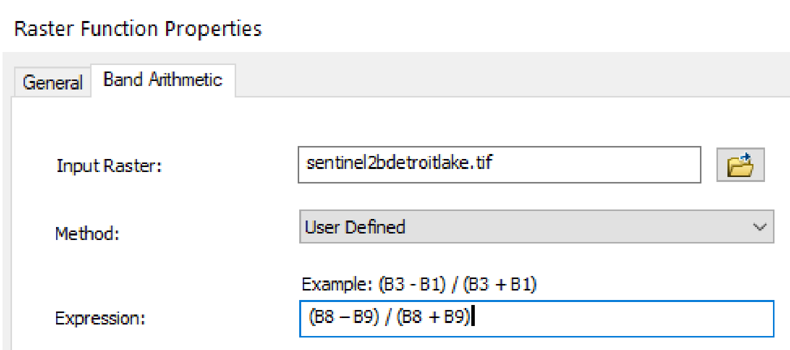


15. Rename the image as RVSI using the same steps from the MSAVI2 and change the symbology of the image to a more colorful scheme.



Q9 [5%]. Include a snapshot showing the RVSI inside the Functions tab and the raster displaying the RVSI using a color scheme different than the default.

16. For the NDWI, you should follow the same steps as for RVSI, with the exception that in the Expression box you should enter the formula for: (B8 – B9) / (B8 + B9).



17. Rename the function computing NDWI and NDVI as “NDWI” and “NDVI”, respectively, using the same procedure as above.

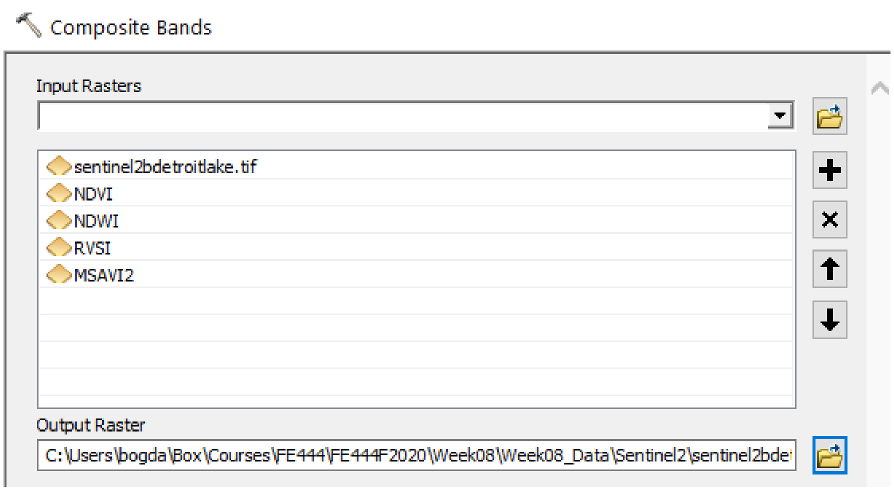
Q10[5%]. Include a snapshot with the Layer Properties showing the NDWI inside the Functions tab and the raster displaying the NDWI using a color scheme different than the default grey ramp.

Now that you have finished computing the four indices, you are ready to create an image that contains not only the reflectance in various wavelengths but also the indices, which will serve as input in the subsequent classification.

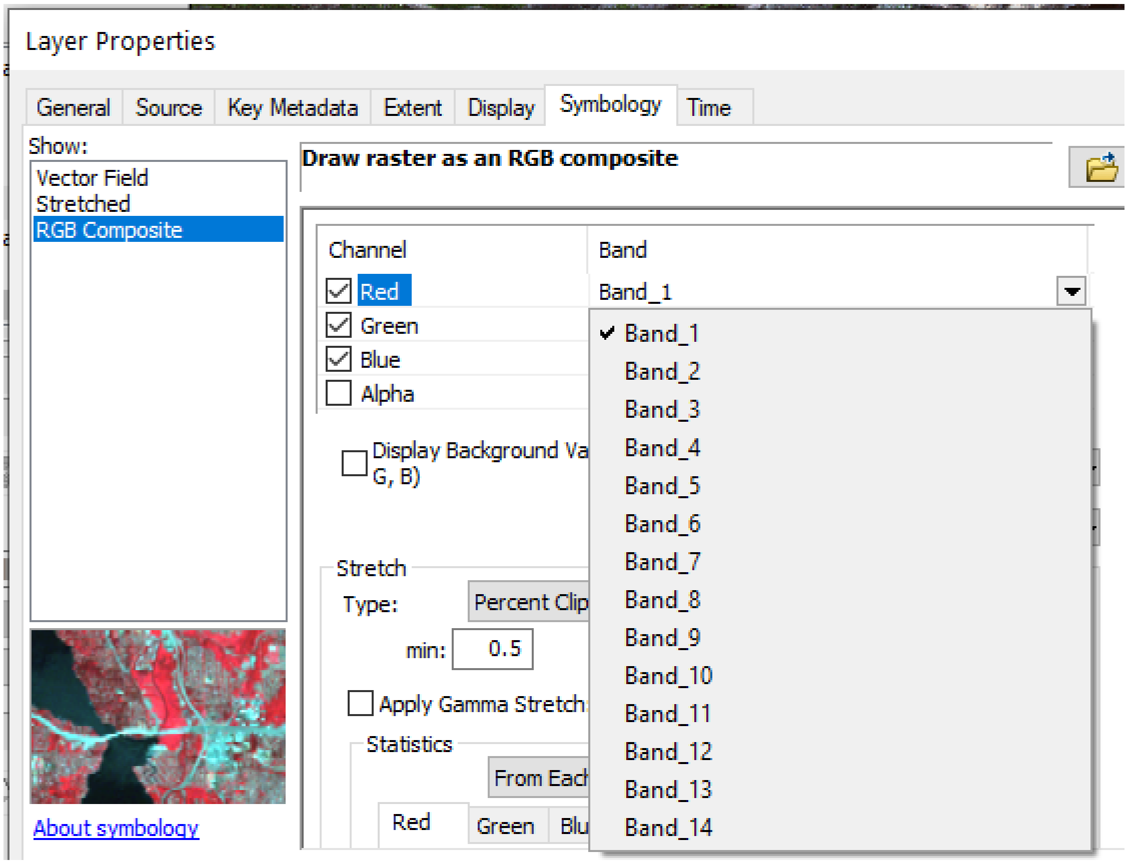
18. To create a multiple band image, you should select from the ArcToolbox the Raster folder, within which choose Raster Processing followed by Composite Bands.

19. After you open the Composite Bands windows, you should select the rasters that will be included in the new image, which are either added to the Input Raster by navigating to the respective files, if they are not in the Table of Contents, or by using the drop-down menu, if they are in the Table of Contents.

20. Because all our rasters are in the Table of Contents, you should add them using the drop-down menu. Be sure that you will sort them such that will make sense later during the classification. For example, it is good practice to have the spectral bands separated from the indices. Therefore, you could place the original 10 bands image first, then add the indices after that. Once you are done, save the new image as sentinel2bdetroitlake14bands.tif.



Q11 [5%] Insert a snapshot with the 14 bands within the sentinel2bdetroitlake14bands image.



8.5 *Classify the image using spectral bands and remote sensing indices*

Execute a maximum likelihood supervised classification and an ISO-DATA unsupervised classification of the sentinel2bdetroitlake14bands image. The two classifications should have the same classes as the classification without the remote sensing indices.

Q12 [10%]. Include a snapshot of approximately 1 square mile for both classifications based on 14 bands, supervised and unsupervised.

Assess the accuracy of both classifications based on the 14 bands image using a confusion matrix.

Q13 [15%]. Include the confusion matrices for both classifications based on 14 bands (i.e., supervise and unsupervised). What is your opinion: did the classification accuracy improved or not by including the remote sensing indices?

8.6 Advanced topics: Multispectral and thermal bands – Remote Sensing Indices

Lab developed by Sudeera Wickramarathna and Dr. Strimbu from Nelson and Khorram, 2018 and ERDAS documentation.

8.6.1 Objectives

- Create binary masks to exclude pixels.
- Create different image indices, including NDVI, NBR.
- Recombining classification components.
- Assessing (qualitatively) the effect of wildfires using image indices.

8.6.2 Files used in this Lab

The files needed to complete this Lab are supplied by the U.S. Geological Survey, and are located on Canvas, Week 8 Lab. Copy the files to a local drive.

Files (.dat and .hdr format)	**Description**
PostFireOLISubset	Landsat 8 OLI bands (seven total), from May 25, 2014
PreFireNBR	Normalized Burn Ratio image from May 9, 2014

The images are related with the wildlife that occurred on May 2014 in San Diego County, CA. In May of 2014, close to 20 different wildfires erupted in San Diego County, triggered by Santa Ana winds and a heat wave. The first fire started on May 5, and the last remaining fires were extinguished by May 22. By May 18, the fires had burned more than 27,000 acres (42 square miles) of land (Figueroa and Winkley, 2014). Some of the communities affected by the fire included Camp Pendleton, Carlsbad, San Marcos, and Escondido.

8.6.3 Background on Burn Indices

Land resource managers and fire officials use burn severity maps from remote sensing instruments to predict areas of potential fire hazards, to map fire perimeters, and to study areas of vegetation regrowth after fires. Landsat imagery has traditionally been used to create indices that indicate burn severity because of its repeated coverage, ease of access, and spectral wavelengths. In this Lab, you will create burn severity images using a variety of different indices, such as burn area index, normalized burn ratio, and normalized burn ratio thermal.

8.6.3.1 Normalized Difference Vegetation Index (NDVI), or greenness map

"The NDVI is a popular tool used in forest vegetation health assessments and represents an index of vegetation greenness. Chlorophyll pigments (light-absorbing pigments) found in the leaves of healthy vegetation absorb incoming radiation (visible light) in the blue (0.45 μm) and red range (0.67 μm) of the electromagnetic spectrum (EMS). Green (0.5 μm) is reflected more so in the near-infrared (0.7–1.3 μm) ranges of the EMS. Particularly in the near-infrared portion of the EMS, there is a strong reflection of light largely due to the internal makeup of a leaf's structure.

When vegetation becomes stressed due to drought, infestation, disease, and other reasons, the leaves typically decrease reflection in the near-infrared range as the leaf's internal structure begins to change. A NDVI image is created for specific image dates through the division of the image's red and near-infrared spectral bands (*NIR – RED/NIR + RED*). Multiple NDVI images can be generated to determine the spectral signatures for areas of healthy vegetation vs. areas of stressed vegetation. This type of application is useful for identifying areas where vegetation is under stress" (Nelson and Khorram, 2018).

8.6.3.2 Normalized Burn Ratio

The Normalized Burn Ratio index highlights burned areas in large fire zones. The formula is similar to a normalized difference vegetation index (NDVI), except that it uses near-infrared (NIR) and shortwave-infrared (SWIR) wavelengths (Lopez, 1991; Key and Benson, 1995).

$$NBR = \frac{NIR - SWIR}{NIR + SWIR}$$

The NBR was originally developed for use with Landsat TM and ETM+ bands 4 and 7, but it will work with any multispectral sensor (including Landsat 8) with a NIR band between 0.76-0.9 µm and a SWIR band between 2.08-2.35 µm.

References

Key, C. and N. Benson. "Landscape Assessment: Remote sensing of severity, the Normalized Burn Ratio; and ground measure of severity, the Composite Burn Index." In *FIREMON: Fire Effects Monitoring and Invenstory System*, RMRS-GTR, Ogden, UT: USDA Forest Service, Rocky Mountain Research Station (2005).

Lopez Garcia, M.J., and Caselles, V. "Mapping Burns and Natural Reforestation using Thematic Mapper Data. *Geocarto International* 6 (1991): 31-37.

Nelson, S.A. and Khorram, S., 2018. *Image Processing and Data Analysis with ERDAS IMAGINE®*. CRC Press.

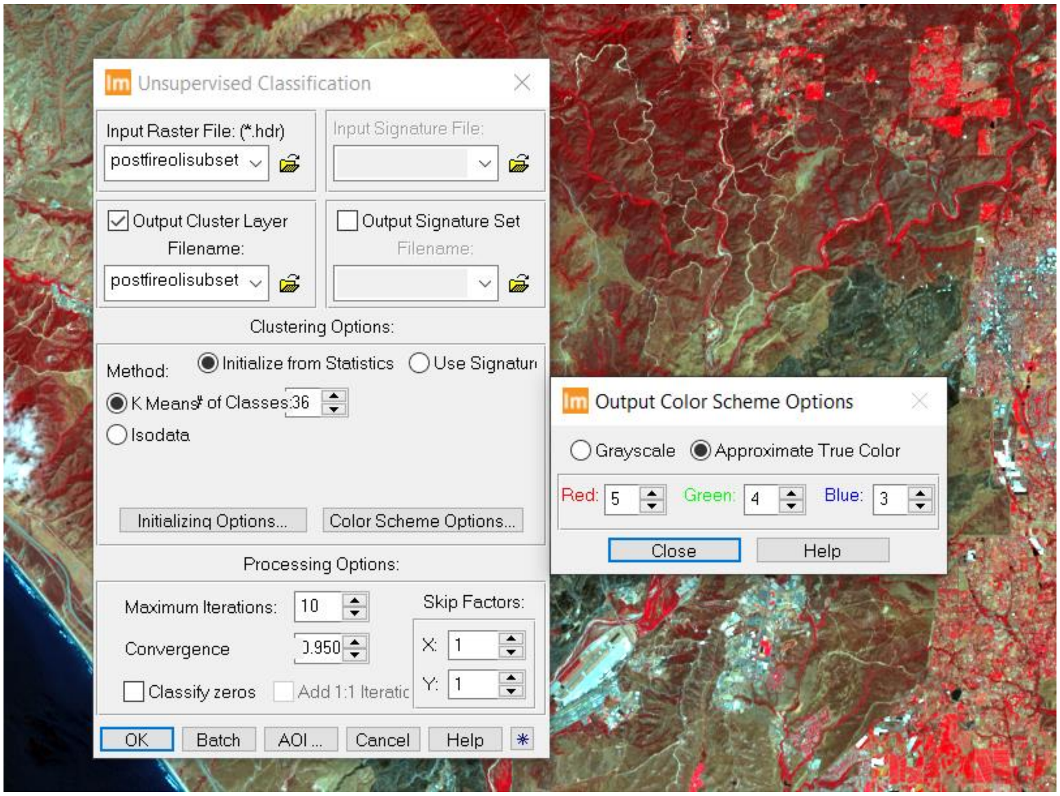
8.6.3.3 Land-Only Image

Generally, the presence of water or water bodies in an image can cause potential issues in land cover classification. Mainly due to the broad spectral range of reflectance values. Therefore, removing the water pixels from the image would help in focusing the classification only on the terrestrial area.

1. Open ERDAS Imagine and click **File → Open → Raster Layer.** Then navigate to the week 8 data folder and select **PostFireOLISubset.** Then display the false-color composite (FCC → Red:5, Green: 4,Blue: 3).

2. Next, we will create a new **unsupervised** classification of the Landsat 8 subset image by selecting **Raster → Unsupervised → Unsupervised Classification**.
3. Specify an output file name of **PostFireOLISubset_unsupclass** in the lab 8 folder. In this lab, we do not need to use the output signature file. Therefore you do not need to check the **Output Signature Set** in the **Unsupervised Classification dialog box.**

4. Now, click on the **Color Scheme Options** button in the Unsupervised Classification dialog box. Change the output color values to 5, 4, 3. You may keep all other settings as default. Then click, **OK**.
5. Next, you will see the process list window and processing of unsupervised classification. Once processing is complete, you can close the process list window.



6. Once the process is completed, you can open the classified image by navigating the **File → Open → Raster Layer.** Then select the **PostFireOLISubset_unsupclass** image.

NOTE: If you have selected the default "**Color Scheme Options → Approximate True Color**" during the Unsupervised Classification, the output color coding of the classified image follows the color scheme of the Landsat 8 FCC subset image. Therefore, the unsupervised classified **PostFireOLISubset_unsupclass** image will look almost similar to the original image.

Q1 (10%). Include a snapshot of the PostFireOLISubset_unsupclass image.

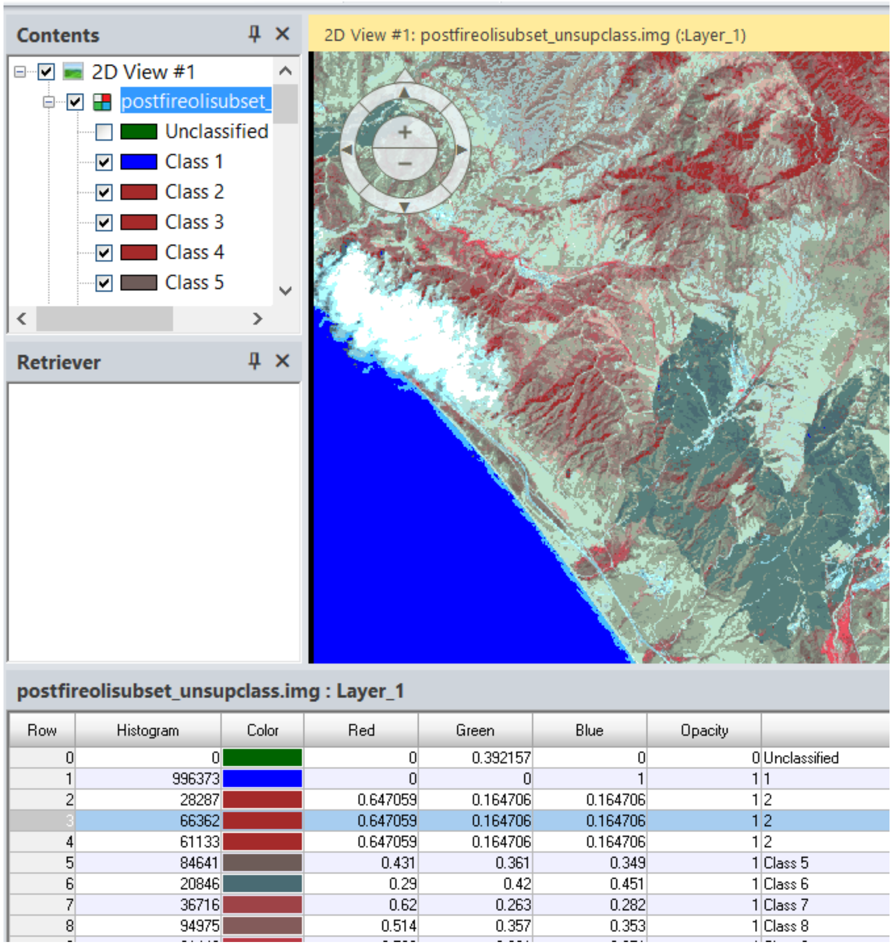
8.6.3.4 Thematic Image Recode

In the next steps, we are going to **recode** the values of classes in an unsupervised classification image. During this process, we will recode the initial classes (i.e., 36 in this case) into two major classes, particularly, either **land** = 2 or **water** = 1.

1. You can right-click on the **PostFireOLISubset_unsupclass** image and select the **Display Attribute Table** option. Now you can explore which classes are water and which are land.

2. Next, you can change the color of each class by clicking on each row in the attribute table to see which classes are "**water**" and which are "**land.**" You can assign blue for water and Brown color for land.

3. Once you have determined the classes belong to water, change them to "1" in the "**Class_Name**" column of the attribute table. Then, you can rename the classes belongs to land as "2" in the "**Class_Name**" column of the attribute table. Do not change the box labeled "**0**" or "**unclassified**".



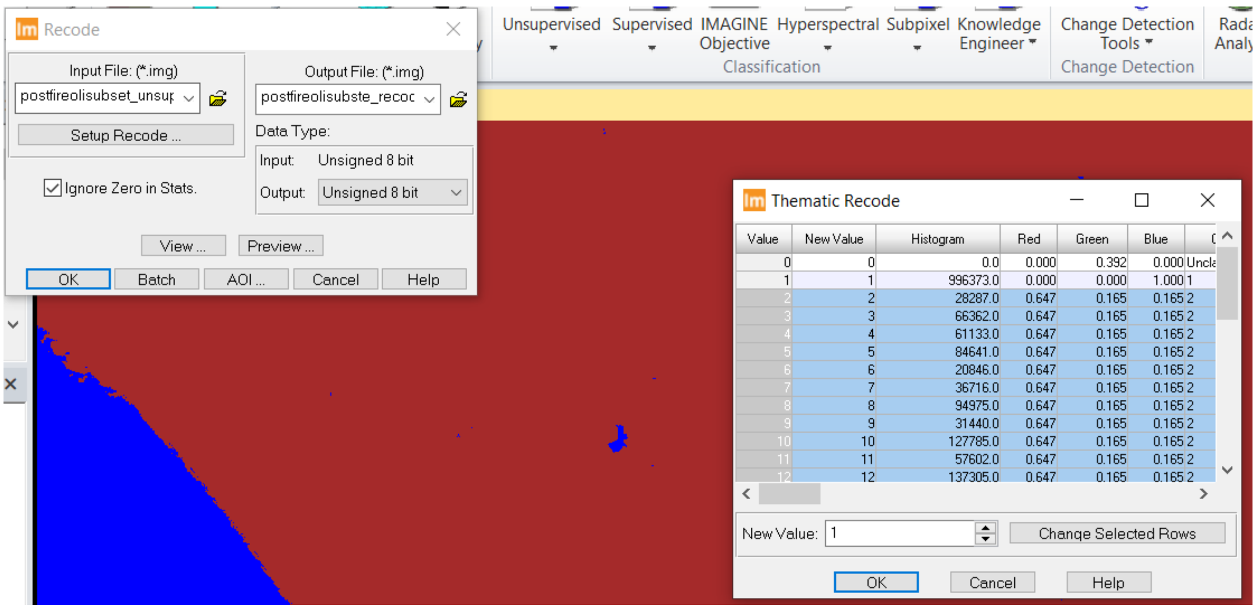
NOTE: You may need to save the work once you changed the classes into water and land. You can perform this by **right-clicking on the file name** in the **Contents Pane** and then **Save Layer** and close the attribute table.

4. Go to **Raster → Thematic** under the Raster GIS category grouping and click **Recode** option. The **Recode dialog** box will open and select **PostFireOLISubset_unsupclass** as the input file. Name the output file **PostFireOLISubset _recode** and select **Ignore Zero in Stats** option. Do not close the Recode dialog window yet.

5. Next, click the **Setup Recode** option. Now we are going to recode the image so that we can differentiate water from land. You can select and highlight the water classes by holding the **Shift key and clicking rows belonging** to the water class **(water=2)**. Next, select **"1"** for **New Value box and click Change Selected Rows.** You will see that changes are updated in the **New Value** column. For example, the **New Value** column will show 1 for all the water classes.

6. Next, you can change the **New Value** column for land classes. You can highlight the land classes using the **Shift key** and then change the number in the **New Value box as 2**. Then select **Change Selected Rows** and click **OK**. Now, you will see that all the classes belong to land converted into **"2"** in the **New Value** column.
Click OK again in the Recode dialog window. Now the new classes will simply be displayed as water = 1 and land = 2.

NOTE: You may need to maximize the **Thematic Record window** so that you can select the rows quickly.



7. Open a second View by clicking **Home → Add View → Display Two Views**. Next, you can open the newly recoded classified image by selecting **File → Open → Raster Layer**. Then select the **PostFireOLISubset_recode** image.
8. Next, we are going to assign the colors to the main two classes: water and land. You can open the attribute table and change the color of class 1 (water) to blue and class 2 (land)to brown.
9. Now, you can compare the recoded image with the original image. For better comparison, you can **link the views and synchronize two views.**

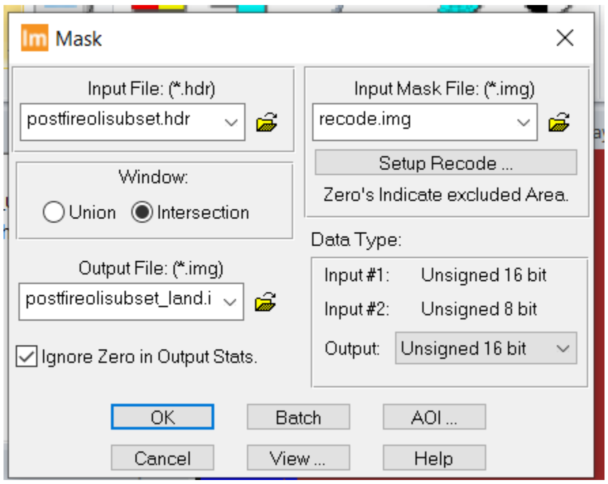
8.6.3.5 Binary Image Mask to Remove Water

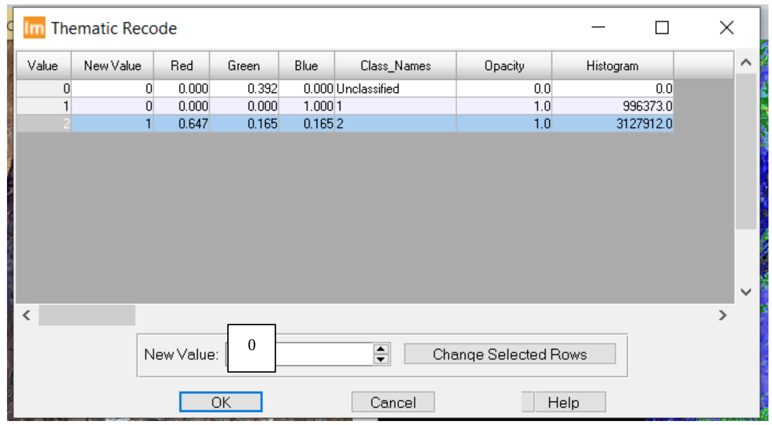
1. Click **Raster → Subset & Chip →Mask** in the **Geometry category group**. Select the original **Landsat 8 subset image (PostFireOLISubset)** as the **input file** and **PostFireOLISubset_recode** as the **input mask file.**

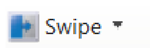
2. Then select **Setup Recode.** Next, click on class "**1**" in the **Value** column to highlight it (Class 1 = water features). Enter "**0**" in the **New Value** box and click on **Change Selected**

Rows. Now you will see the **New Value** column with the updated value of 0 for water class.

3. Similarly, you can click on class "**2**" in the **Value** column to highlight it. Then change class 2 to a value of "**1**" (Class 2 =land features) in the **New Value** box and click on **Change Selected Rows.** Now you will see the **New Value** column with the updated value 1 for land class.
4. Name the output file **PostFireOLISubset_land**. Make sure to activate **Ignore Zero in Output Stats** option and click **OK.**

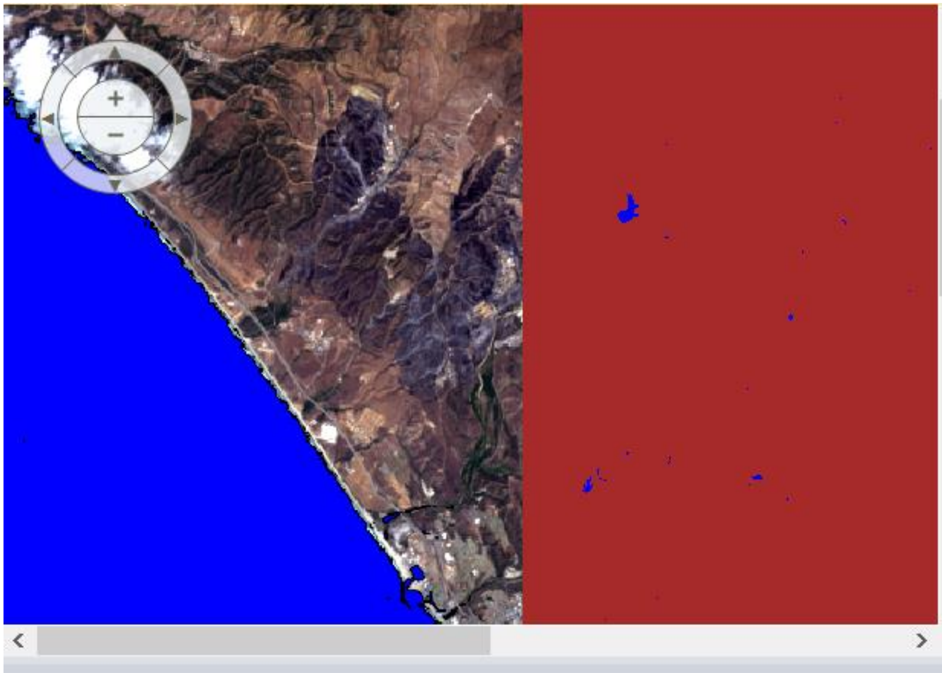




5. Next, add the new **PostFireOLISubset_land** file to **2DView#2** and make false-color composite as (Red: 5, Green: 4, Blue: 3) to compare the output, you can do this by swiping between **PostFireOLISubset_recode** and **PostFireOLISubset_ land** files. Use the **Swipe option** under **Home → Swipe** 

. Now you have created a classification mask image that contains only the land features, as the water features are now *masked out*.

Q2 (10%) Insert the screenshot of your masked out image.



8.6.4 Create a Normalized Difference Vegetation Index

From previous steps, we have removed the water features from the image. Now we are going to perform a specialized type of unsupervised classification on just the land area. *"This specialized type of unsupervised classification is a vegetation index representing the amount of greenness within the image and is known as a Normalized Difference Vegetation Index or NDVI."*

1. First, clear the **2D View#1** and select **Raster → Unsupervised → NDVI** under the **Classification category group**. Use **PostFireOLISubset_land** as the input file and name the output file **PostFireOLISubset_ndvi**.

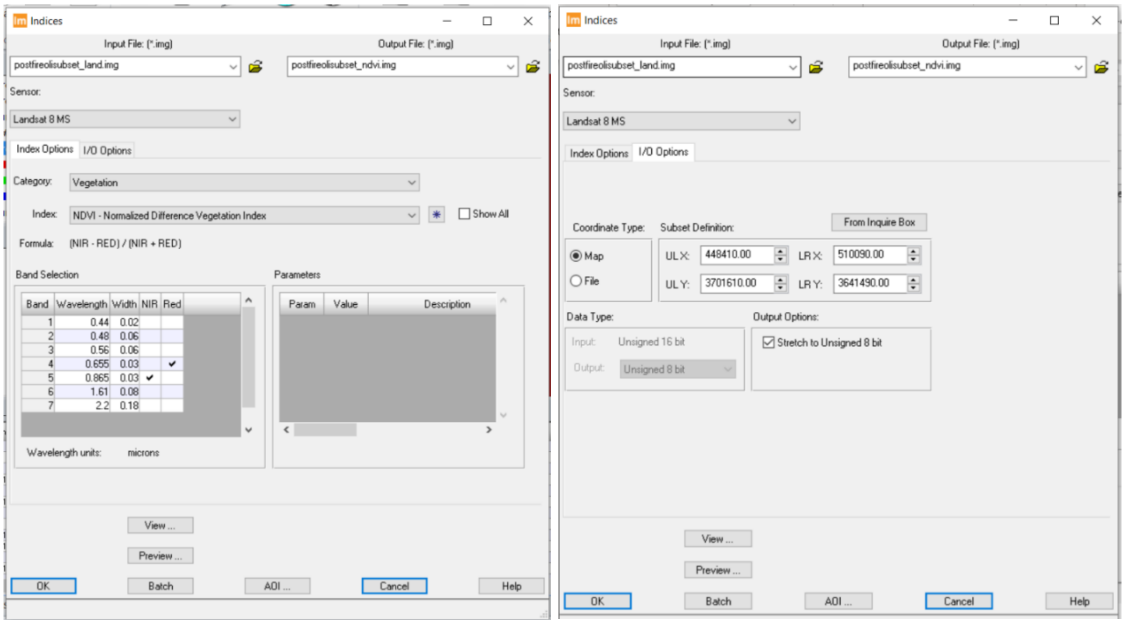
NOTE: Remember, the **PostFireOLISubset_land** file contains only the land features, as the water features were masked out in the previous steps.

2. Next, make the following selections in the Indices dialog window: select **Senor as Landsat 8 MS** and **Category as**Vegetation.

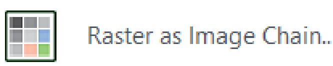
NOTE: The image that the **PostFireOLISubset_land** image was originally derived from the Landsat 8 sensor. Because this option does not show up, it is necessary to ensure the correct bands that relate to the Landsat 8 (Red (Band 4) = 636–673 nanometers and NIR (Band 5) = 851–879 nanometers) are specified (Nelson and Khorram, 2018).

Useful link: https://www.usgs.gov/core-science-systems/nli/landsat/landsat-8
3. **Index:** NDVI—Normalized Difference Vegetation Index as the function should be: **(NIR - RED)/(NIR + RED).**

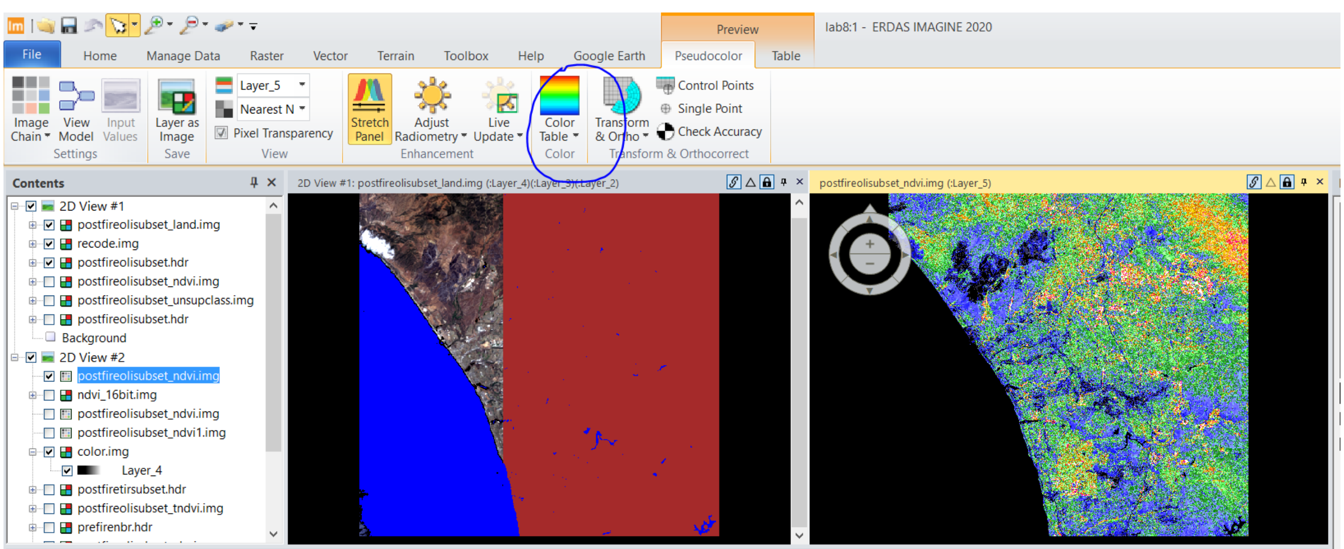
Band Selection: In the NIR column check Band 7, and in the Red column check Band 5.
4. Click the **I/O Options** tab and select **Map** as the coordinate type and check. You can leave other settings as default.



5. Select **2D View#2** and then go to **File → Open → Raster as Image Chain..**



. Navigate to your folder and open the newly created **PostFireOLISubset_ndvi**.

6.


You will notice that the new **Preview|Pseudocolor** toolbar will open.

7. You can click the **Image Chain** icon  and then select **Pseudocolor.** Then use the options available under **Color Table** to visualize the NDVI image.

NOTE: Based on your interest or purpose, you can select the options under **Color Table.**

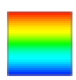
1. Raster as Image Chain

You can apply enhancements, apply color ranges, and extract features from raster images using the Raster as Image Chain option. When you open a Raster image using the **Raster as Image Chain** option, one of the following tabs becomes available depending on which image chain is currently selected.

2. Pseudocolor tab contains tools for applying a color table onto grayscale, elevation, and thematic images in addition to single layers of multispectral images after their stretches are applied. It is generally intended for use with data that is ordinal in nature, such as digital elevation models (DEMs), brightness / reflectance / radiance imagery, thermography, and so forth, so that you can apply a color ramp to help you visually interpret data that would otherwise be displayed as grayscale.

This tab is available when you use the Pseudocolor image chain to view your image. This happens by default when you open a thematic Raster that does not have a color table using either **File > Open > Raster as Image Chain** or through connecting a thematic Raster output that does not have a color table with the Preview operator in the Spatial Model Editor. This tab also opens when you select the Thematic image chain from the **Image Chain** menu on another Image Chain tab.
The **Pseudocolor** tab is grouped into these categories (see below for details):

- Settings
- Save
- View
- Enhancement
- Color

3. **Color Table -** Open the Color Table  gallery as well as the Color Ramp Panel. Note that any stretches are applied prior to applying the color ramp mapping. Consequently any input data range can still be mapped through a color ramp which might only consist of 256 colors. This is possible because the stretch, such as a 2 Standard Deviation stretch, maps the input data into the range used by the color table. This also means that altering the stretch will alter the distribution of colors that are applied and can be used to assist in interpreting the data visually. (source: ERDAS Imagine).

Q3 (15%): As mentioned earlier, this PostFireOLISubset image was captured after a wildfire and we use that image to derive PostFireOLISubset_ndvi. Now, select an appropriate option (color ramp) from Color Table to detect the wildfire patches using PostFireOLISubset_ndvi. Insert a screenshot of your screen which has the orginal PostFireOLISubset image and PostFireOLISubset_ndvi image. (similar to the above figure).

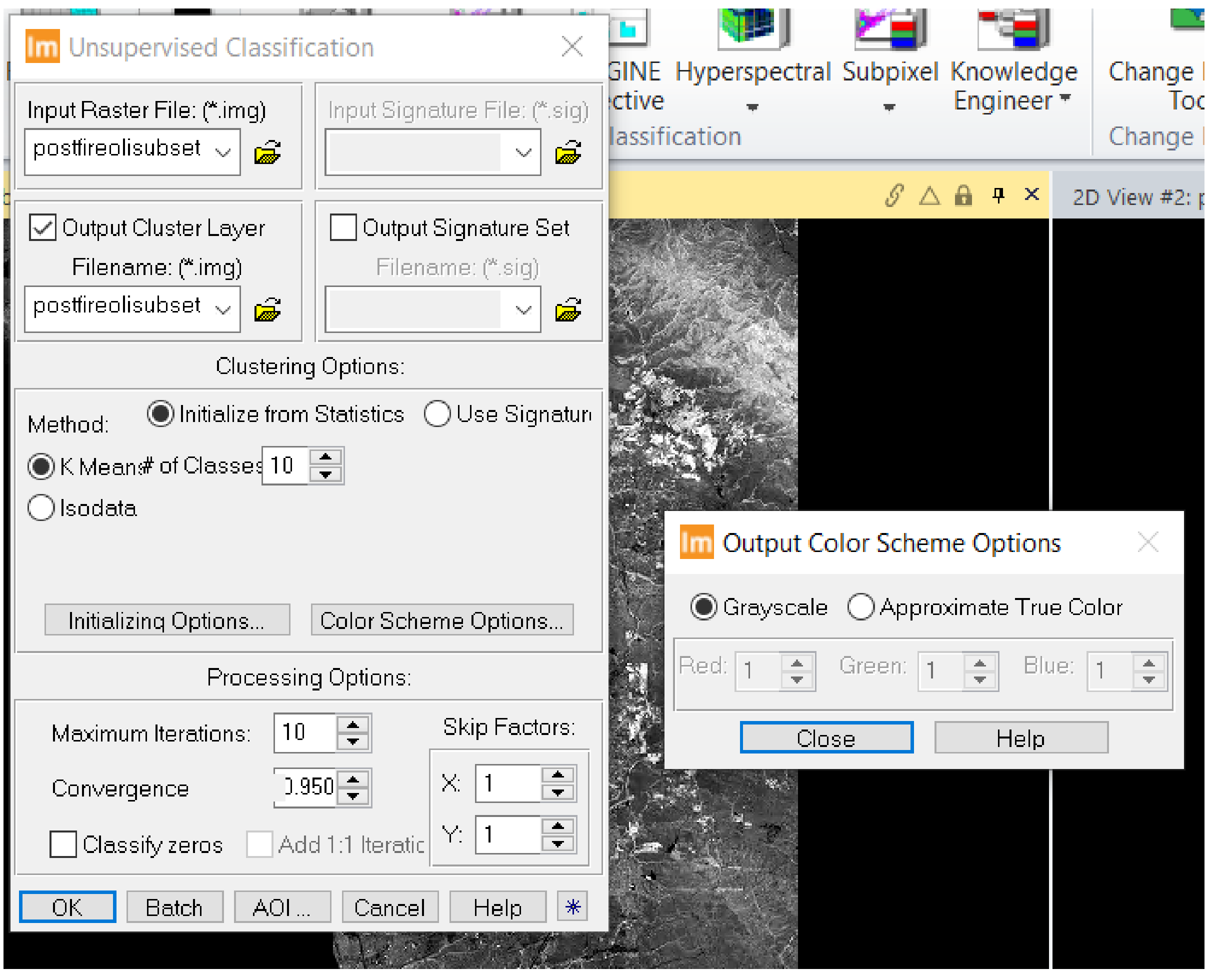
Q5 (5%) Write a short description of how you delineate or estimate the impact (area) of wildfire using ERADS Imagine options you learned so far.

8.6.5 Classify the image using Normalized Difference Vegetation Index

Next, you are going to perform an unsupervised classification on the previously created **PostFireOLISubset_ndvi.**

8. Now select **Raster → Unsupervised→ Unsupervised Classification** option available under the **Classification category group**. Select **PostFireOLISubset_ndvi** as input image and select **PostFireOLISubset_veg** as the output file name.

9. Then specify the number of output classes and select 10 for classes for this exercise. Keep the other settings as in the following figure (settings). Next, select **Color Scheme Options** and choose **Grayscale**. Also, the output signature file is no need for this exercise. So you can uncheck the box for that option and click **OK.**



10. Next, the Process List window will appear once the processing is complete. C**lose** the process list window.

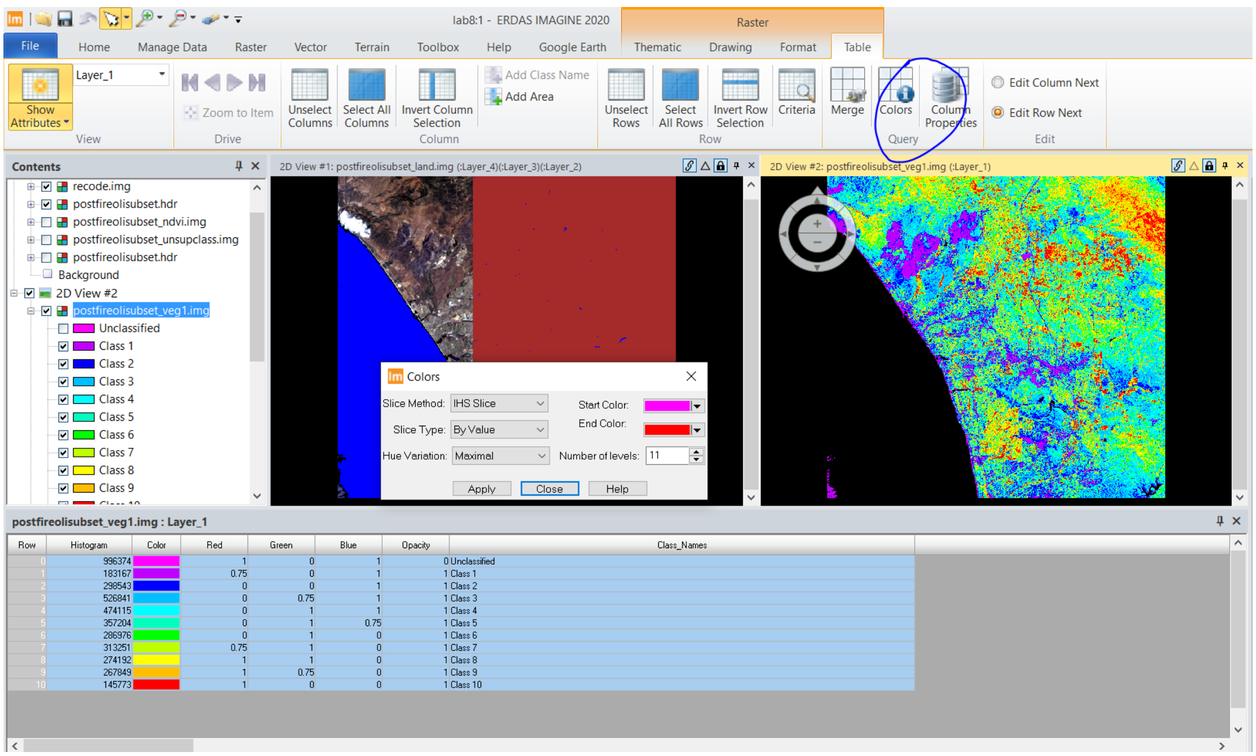
11. Now you can open the newly classified image by the following **File → Open → Raster Layer** and select the **PostFireOLISubset_veg** image.

12. Right-click on the **PostFireOLISubset_veg** in the content pane and open the attribute table. You should see 10 classes and one **unclassified** labeled as **0**.

13. You can change the colors of classes 1–10 and try to differentiate between different vegetation classes. You should be able to separate a minimum of five classes: For example, you can select classes such as **forest** (**deciduous forest** or **evergreen forest)**, **herbaceous grass**, **burned areas, developed (non-vegetated)**, and **barren land (mostly beech areas)**.

14. Think and decide whether if you need to combine more classes for better classification output.

NOTE: **Alternatively, you can try** Raster > **T**able> Colors **in** Query group **to assign color ramp. You can select** Start Color and End Color**; ERDAS will create a color ramp for you.**



15. In this exercise, you are not required to perform an image classification that you have done in Lab 6 or Lab 7. But if you are interested, you can perform a detailed image classification as we have done in Lab 7, by recoding the 10 classes into 5 classes.

16. To see just the output, you can assign similar colors to the classes that you want to combine, as explained in the following images:

Save your work frequently!

postfireolisubset_veg1.img : Layer_1

Row	Histogram	Color	Red	Green	Blue	Opacity	
0	996374		1	0	1	0	Unclassified
1	183167		0.75	0	1	1	Burened forest
2	298543		0	0	1	1	Class 2
3	526841		0	0.75	1	1	Urban
4	474115		0	1	1	1	Urban
5	357204		0	1	0.75	1	Class 5
6	286976		0	1	0	1	Class 6
7	313251		0.75	1	0	1	Class 7
8	274192		1	1	0	1	Class 8
9	267849		1	0.75	0	1	Class 9
10	145773		1	0	0	1	Class 10

Row	Histogram	Color	Red	Green	Blue	Opacity	
0	996374		1	0	1	0	Unclassified
1	183167		1	0.647059	0	1	Burned forest
2	298543		0	0	1	1	Urban
3	526841		0	0	1	1	Urban
4	474115		0.498039	1	0	1	Decidous forest
5	357204		0.498039	1	0	1	Decidous foest
6	286976		0.627451	0.321569	0.176471	1	Barren Land
7	313251		0.627451	0.321569	0.176471	1	Barren land
8	274192		0	0.392157	0	1	Evergreen forest
9	267849		0	0	1	1	Urban
10	145773		0	0.392157	0	1	Evergreen forest

Q6 (15%). Include screenshots of the attribute table with combined classes (as above figure) and the classified image.

8.6.6 Create an Impervious Surface Map (Remove Vegetation)

Since we have completed the NDVI output, we can now continue our work to determine if the developed areas (such as impervious land cover) can be better isolated by removing vegetation features.

8.6.6.1 Create an Impervious Surface Map (Remove Vegetation)

1. This section, we are going to mask out all vegetative features while leaving only the developed features remaining in the image.

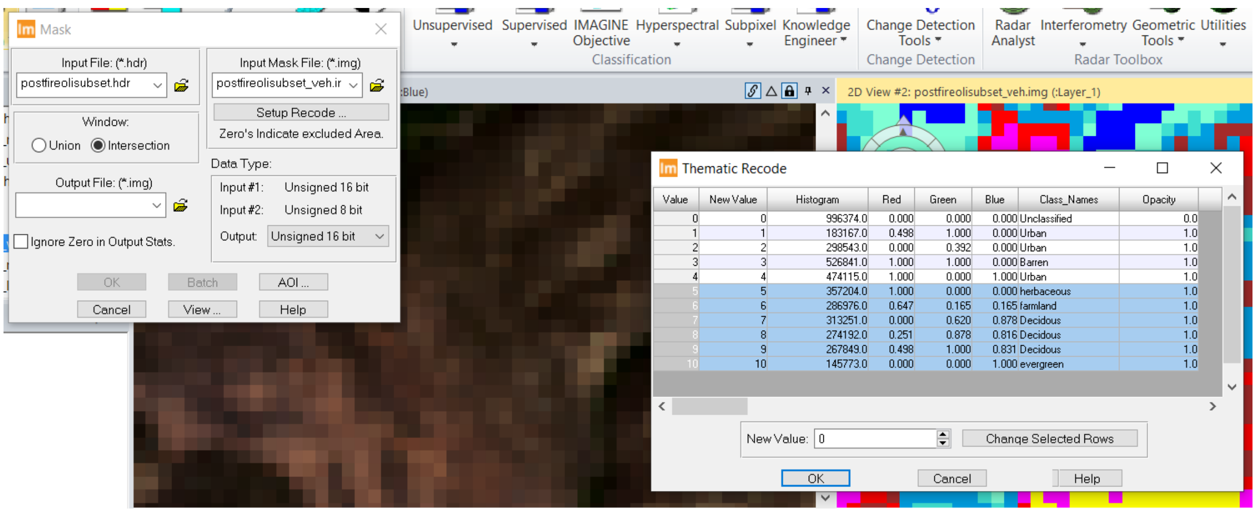
2. First, go to **Raster → Subset & Chip → Mask**. Then Mask dialog box will open and select the **PostFireOLISubset** image as the input file. Then select **PostFireOLISubset_veg** as the input mask file.

3. Next, click on **setup Recode** and highlight all the vegetative classes (i.e., deciduous, evergreen, and farmland…etc.).

4. Now, enter "**0**" in the **New Value box** and then click **Change Selected Rows.** Similarly, highlight all the developed classes/non-vegetation features (i.e., developed and barren) and enter "**1**" in the **New Value box** and click on **Change Selected Rows**. Once you finished this step, you will see the updated values in the **Value** column.

5.You can name the output file as **PostFireOLISubset_devmask**. Also, select **Ignore Zero in Output Stats** and select **OK** to create the mask.

6. Next, you can see the new **PostFireOLISubset_devmask** by adding it to the 2D View#2. Try to swiping between **PostFireOLISubset_devmask** and **PostFireOLISubset_veg.**



Q7 (10%). Include a Screenshot of PostFireOLISubset_devmask image with PostFireOLISubset_veg underneath (use Swipe tool, so you can see both the images)

8.6.6.2 Classify the Impervious Surface Map

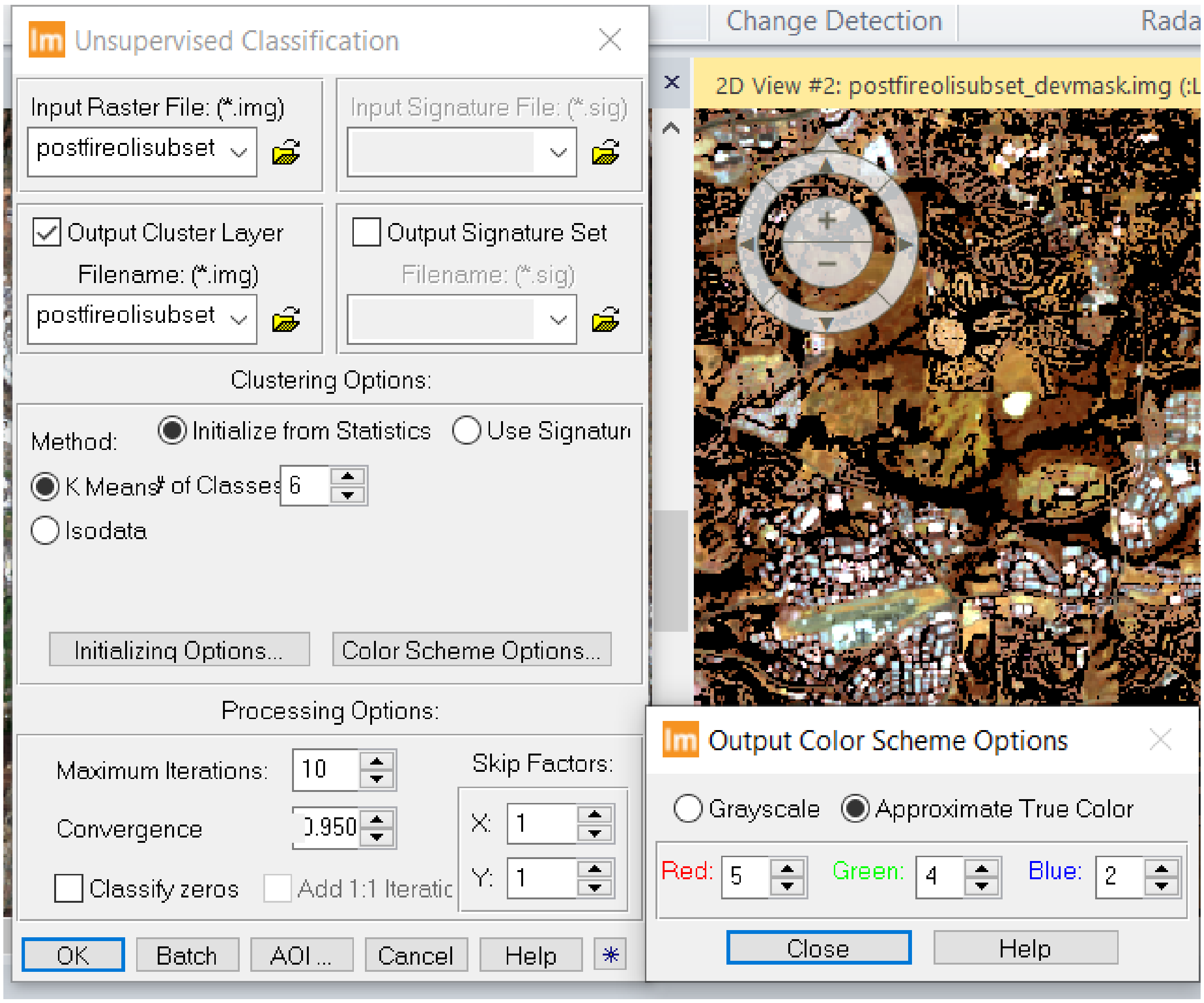
In this section, we are going to perform unsupervised classification just considering the developed or non-vegetated categories. We will be using an unsupervised classification and the mask created in the previous step.

6. First, we have to choose a false-color band combination for displaying the **PostFireOLISubset_devmask** image you just created. This step will help you to separate different intensities of development or types of impervious surfaces (i.e., 5, 4, 2).

7. Then go to **Raster → Unsupervised → Unsupervised Classification** under the Classification category group. Select input file as **PostFireOLISubset_devmask** and name the output file name as **PostFireOLISubset_impervious**.

8. Next, you can specify the number of output classes. For this exercise, choose 6 (for low, medium, and high-intensity urban cover) and uncheck the output signature file box.

9. You need to adjust **the Color Scheme Options → Approximate True Color** to match the band combinations (e.g., Red:5, Green: 4, Blue: 2) and click **OK.**



10. A Process List window will appear. Within a short time, processing will complete and you canclose the process list window.

11. You can open the newly classified image by selecting **File → Open → Raster Layer** and select the **PostFireOLISubset_impervious** image.

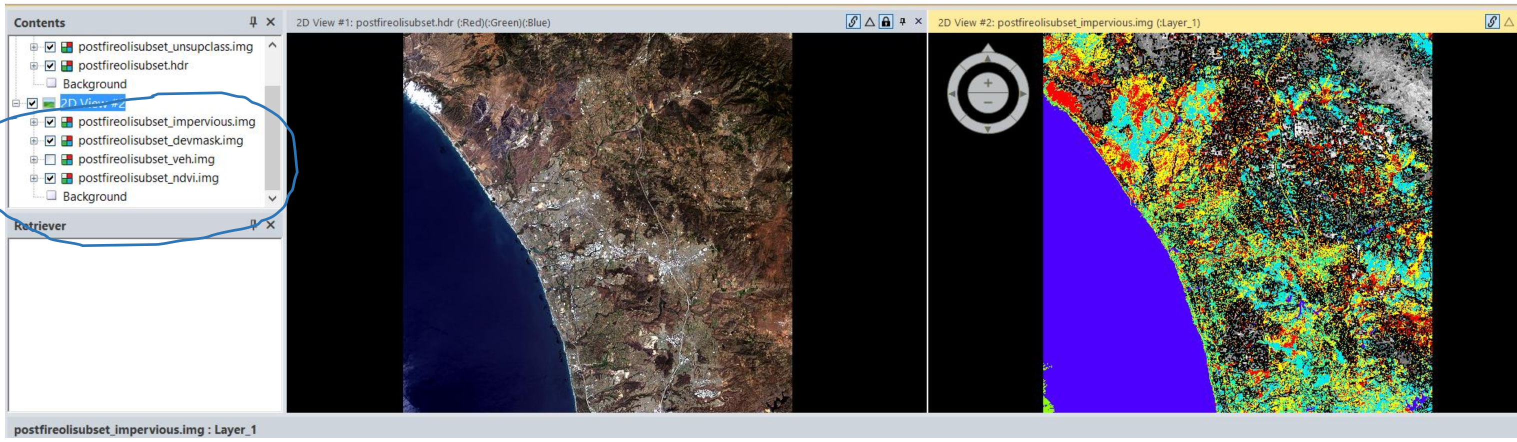
12. Now you can zoom into the image to observe the different class of impervious cover. You can change each classes color in the attribute table if needed.

Q8 (10%). Include a screenshot of your classified PostFireOLISubset_impervious image.

8.6.6.3 Recombine Classification Components

Under this section, we will recombine the three classification components in the viewer (i.e., water-only, NDVI-veg, and other-Impervious). This will provide a final output with a more detailed classification.

1. You can just try to visualizing a combined dataset by selecting **PostFireOLISubset_unsupclass**, **PostFireOLISubset _veg**, and **PostFireOLISubset_impervious** in the 2D View#2 window.

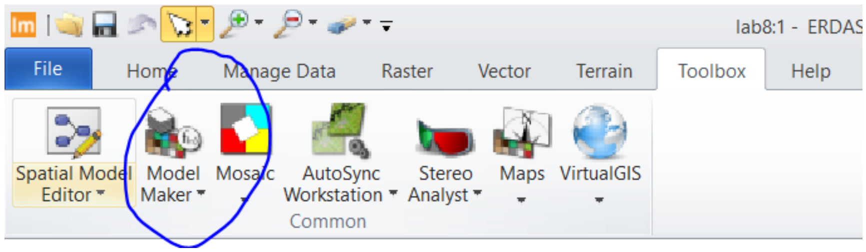


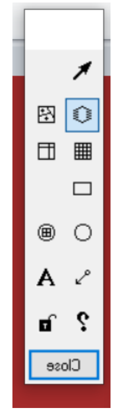
8.6.7 Task automation in ERDAS: Model Maker

In many instances you are faced with creating of a succession of tasks that most likely will be repeated. In this case Model Maker tool can be used in ERDAS. Model Maker is a spatial modeler language that is presented as a graphical interface rather than as a code. Model Maker allows you to develop customized algorithms (succession of tasks) by recoding each output layer and then combining them. For example, suppose the analyst wants to create an image to strongly accentuate impervious features possibly. In that case, the **PostFireOLISubset_impervious** image (containing only impervious features masking out all other non-impervious features) could be added to the recoded the **PostFireOLISubset_veg** file containing all 10-land cover class, plus the unclassified class.
The resultant image would produce a combined image that has the impervious-only layer added into the vegetation recoded original image, which also contains the impervious data, thus *theoretically* increasing/improving the impervious signature (Nelson and Khorram, 2018).

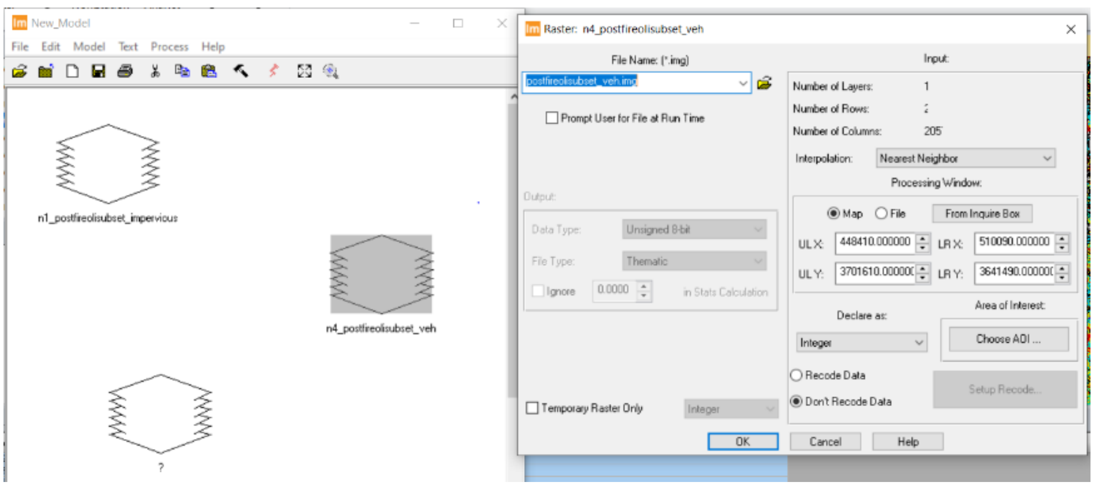
The following steps will perform this operation with Model Maker:

1. First, you need to start Model Maker by navigating to **Model Maker → Toolbox** tab.

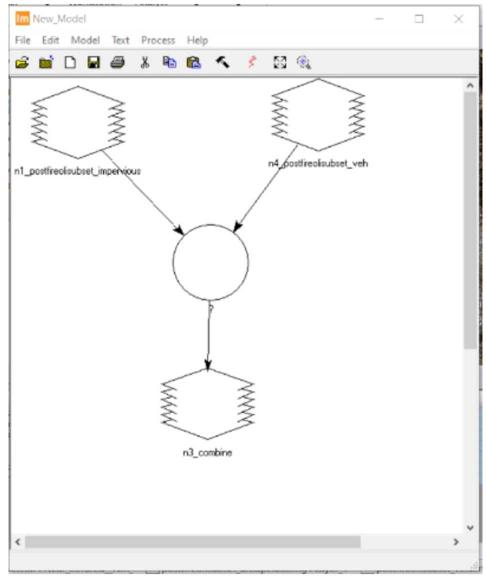




2. Next **New Model** dialog box will open. Select the **Place a Raster object in the model** object in the model space. Add this object three times, as shown in the following figure.



3. Now, you need to double-click on each Raster Object to specify the input layers (**PostFireOLISubset_impervious** and **PostFireOLISubset_veg**), as well as name the output image layer (such as **PostFireOLISubset_combined_veg_imperv or combine).**
4. A raster dialog box will open, and you can enter your respective image file each time, as shown in the above figure.



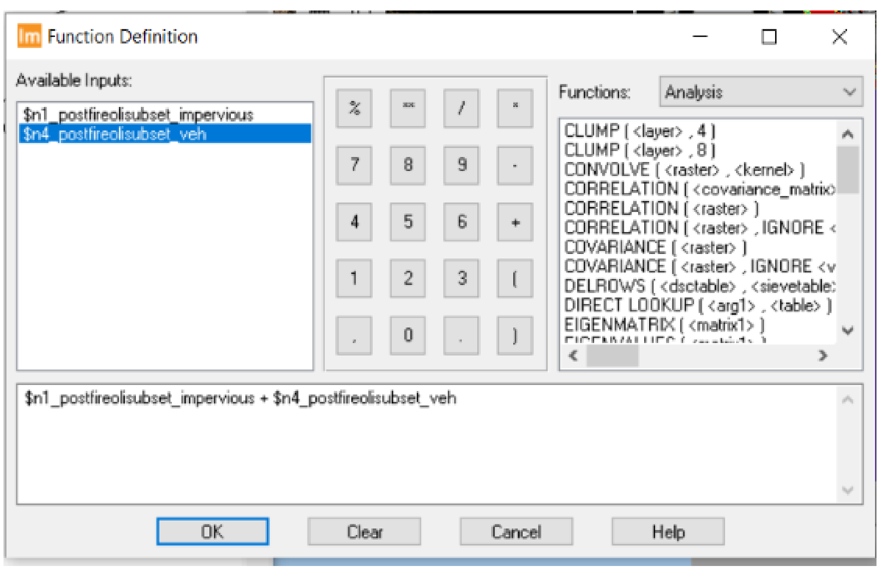
5. Then, you have to select the **Place a function in the model** option to connect the raster objects. You can use the cursor to draw a function circle in the middle of the model space.

6. Next, select the **Connect inputs to function or function to outputs** option to connect the two input image layers (**PostFireOLISubset_impervious** and **PostFireOLISubset _veg**) to the function as shown in the image.

Q9 (10%). Insert a screenshot of your model (similar to the above figure)

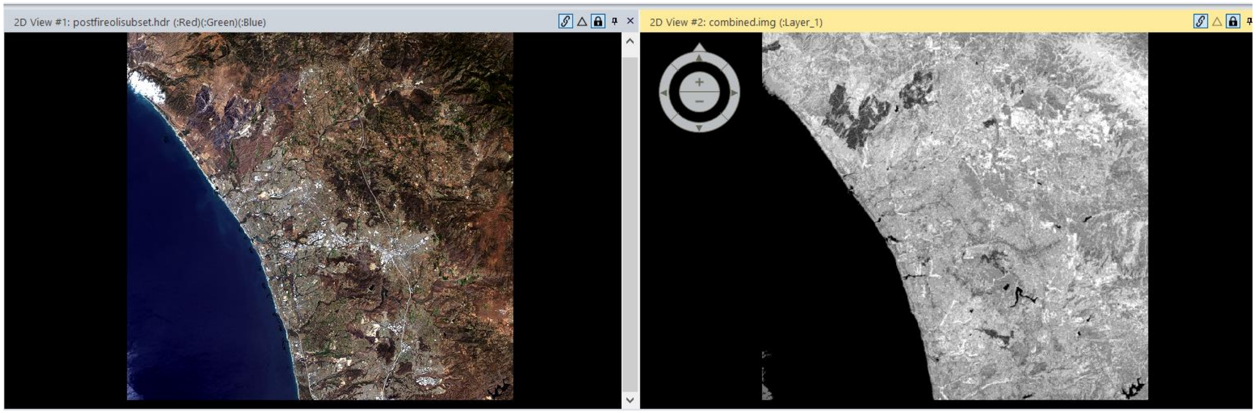
Also, add a single arrow to connect the function to the output raster object (**PostFireOLISubset_combined_veg_imperv or combine**)

7. Double-click the function circle in the model space. Then the **Function Definition** dialog box will open. In the **Function Definition** dialog box, you can select **Arithmetic** from the **Function** dropdown list and create the expression:
$n2_ PostFireOLISubset _veg + **$n1_ PostFireOLISubset _impervious**.

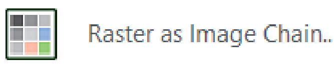


8. Then click **OK** in the **Function Definition** dialog and click the red lightning bolt in the **New Model** window to run the model.

9. Once the Process List is completed, you can close this list and the New Model window. Then open the new image layer in ERDAS Image. You may compare this image to the original **PostFireOLISubset_impervious** image.



NOTE: If you want to add the colors to the output, you can follow the following steps. Select 2D View#2 and then go to **File → Open → Raster as Image Chain..**

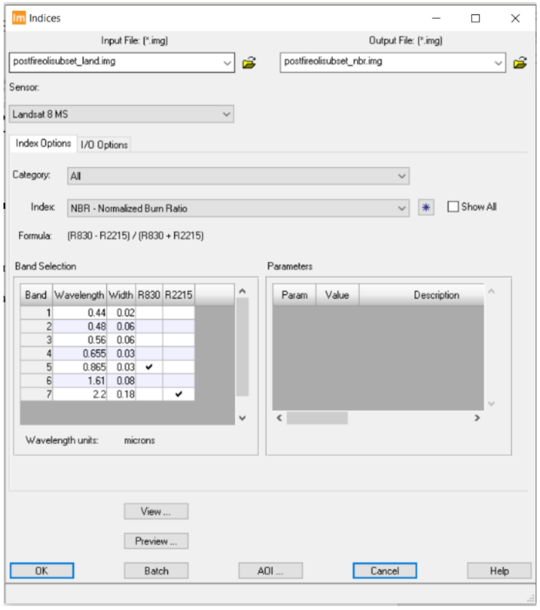


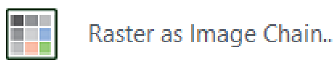
. Navigate to your folder and open the newly created **PostFireOLISubset_ndvi**.

Q10 (15%) Include the output of your PostFireOLISubset_combined_veg_imperv image.

Extra Credits (20%): Working with NBR- Normalized Burn Ratio index

1. First, select **Raster → Unsupervised → Indices** under the **Classification category group.** Use **PostFireOLISubset_land** as the input file and Choose the **sensor as Landsat 8MS.**
2. Select the Index as **NBR-Normalized Burn Ratio** and name the output file as **PostFireOLISubset_nbr**. You can keep the other settings as default (refer to the following figure) and click **OK. Now open the image using the following steps.**



3. Select 2D View#2 and then go to **File → Open → Raster as Image Chain..** 

. Navigate to your folder and open the newly created **PostFireOLISubset_nbr**.

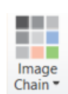
4. You can click the **Image Chian** icon 

and then select **Pseudocolor. Then** use the options available under **Color Table** to visualize the **PostFireOLISubset_nbr** image.

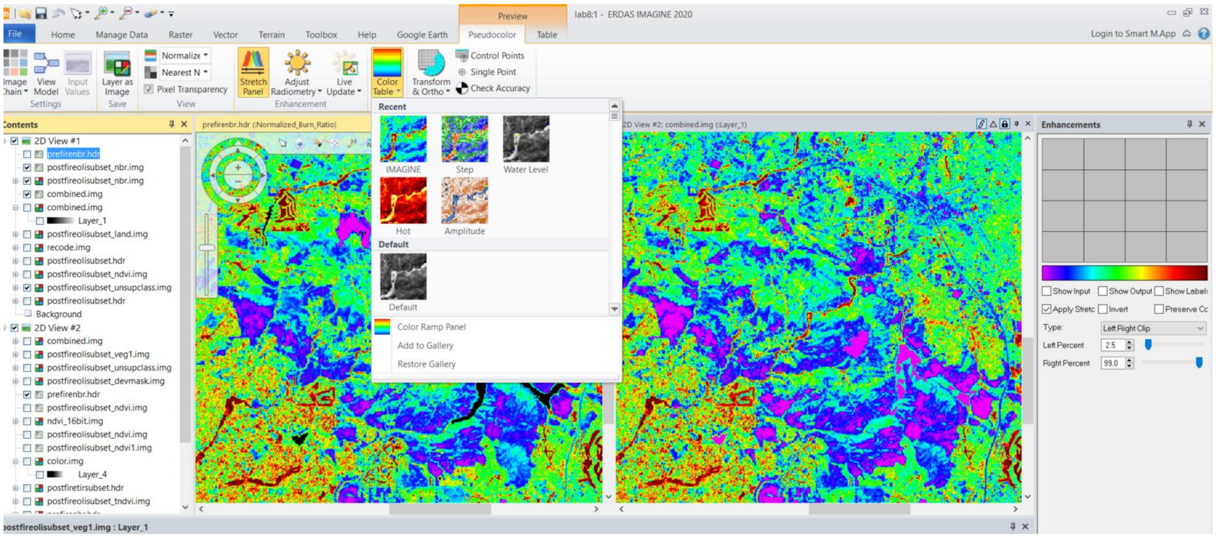
5. **Compare PostFireOLISubset_nbr vs. PostFireOLISubset_ndvi,** you created either using two View windows or Swipe tool (You have to use the same color ramp for this step).

EC1 (10%). Describe with few steps how you can use this approach to determine (qualitatively) the damage that occurred due to the wildfire.

6. Now, navigate to the data folder and add the **Prefirenbr** to the 2D View#2 window by using the following above step 3.

7. You can click the **Image Chian** icon 

and then select **Pseudocolor. Then** use the options available under **Color Table** to visualize the **Prefirenbr** image. When you select a color table, select **IMAGINE color ramp from Color Table for both the images.**



NOTE: **Prefirenbr** is Normalized Burn Ratio image from May 9, 2014.
PostFireOLISubset_nbr is Normalized Burn Ratio image you just created from Landsat 8 OLI bands (seven total) from May 25, 2014.

EC2 (10%). Now you have Normalized Burn Ratio images from two dates to compare. Think of a possible method that you can implement to estimate the spread of wildfire from May 9 through May 25, 2014. (You can answer this with a few sentences)
Hint: Estimating the burned area by demarcating the burned patches.

9 Introduction to Lidar

Lab developed by Dr. Strimbu from USFS documentation.

9.1 Objectives

- Understanding and accessing the information stored in a las file
- Basic processing of point cloud
- Introduction to Fusion, the USFS lidar processing software

9.2 Files used in this Lab

The files needed to complete this Lab are supplied by the U.S. Forest Service, and are located on Canvas under Week9 module.

Files	**Description**
lda_4800k_data.las	Point cloud
Orthophoto_4800k.jpg	Ortho-rectified image
Orthophoto_4800k.jpgw	Ortho-rectified image georeferenced
4800k_DSM	Digital surface model
4800k_ground_surface	Digital terrain model
three_plots.shp (shx, dbf etc)	Shapefile

Lab Content

9.3 Part I: Introduction to Fusion

9.3.1 Download and Install Fusion (this step may not be necessary if you are in the lab or access Fusion through Citrix).

9.3.2 Download Fusion

Open a Browser

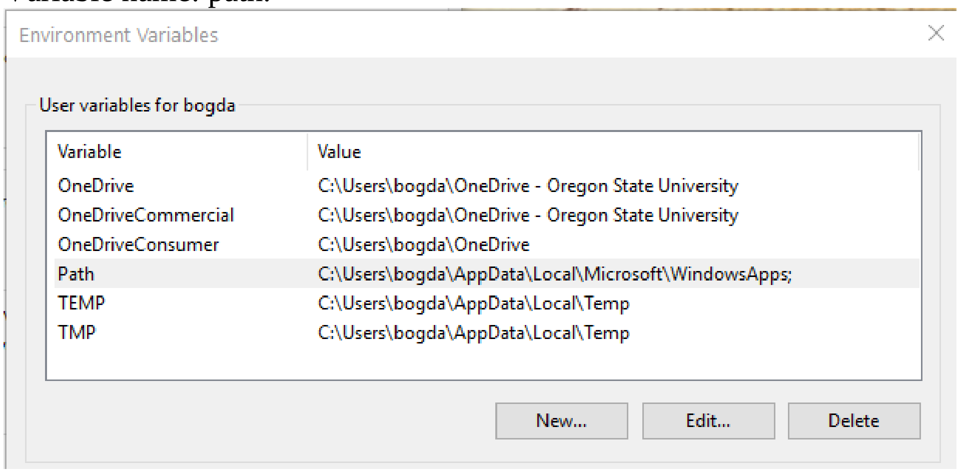
1. Connect to: **http://forsys.sefs.uw.edu/fusion/fusionlatest.html** to open "The Latest Version" Fusion webpage.
2. Click the Install File hyperlink to open the File Download dialog for **Fusion_Install.exe**.
3. Click the Save button to open the Save As dialog.
4. Navigate to a location of your choosing (*one that you will remember*) and/or create a new folder and click Save. The file (**Fusion_Install.exe**) will download.

When the download is complete, Open (or Run) Fusion_Install.exe.

1. The first screen of the Fusion Setup dialog will display.
2. Keep the default installation Components and click Next
3. Keep the default Install Location and click Next
4. And, keep the defaults for the Start Menu Folder and click Install.
5. After the installation completes, click Close.

9.3.3 Add Fusion.exe to the Environmental Variables

1. Open the Start Menu and Right-Click on My Computer or This PC, depending on the Windows version.
2. Select Properties from the drop-down menu and then click Advanced system settings in the left sidebar.
3. In the System Properties dialog box select the Advanced tab.
4. Under the Advanced tab click the Environment Variables button at the bottom.
5. In the Environmental Variables dialog box, under “User variables for (your username)” (see following graphic), if there is a path variable click Edit. If not, click New and call the Variable name: path.



6. For Variable value, enter the directory for Fusion.exe, (ideally c:\FUSION) and click OK.
7. Click OK in the remaining dialog boxes to close them and retain the edit. Now let’s quickly test the new environment variable.
8. Open a Command Prompt by clicking Start > Programs > Accessories > Command Prompt and type catalog and press Enter. If you are using Windows 10, you can type “Command Prompt” in the search bar, then press Enter. You should now see a description of the Fusion command line executable “catalog.exe”.
9. If the command prompt did not recognize catalog as a command, the edits you made to the path environmental variable were not successful. If this is the case, you can repeat the above steps and try again, ask the course instructor or TA. You may need administrative privileges to edit the user Environmental Variables.

9.3.4 Download LAStools

1. Connect to the LAStools site: https://rapidlasso.com.
2. On the main menu, click LAStools > download.
3. Unzip the downloaded file and navigate to LAStools > LASzip > dll.
4. Copy the LASzip.dll file.
5. Navigate to the folder where you installed Fusion and paste the LASzip.dll file in there

9.4 Getting started with Fusion and Loading the Example Lidar Data

A. Click the shortcut to start Fusion

1. Click the Raw Data button on Fusion toolbar to display the Open dialog box.

2. Navigate to your data folder (for example, C:\lidar\SampleData\) and Select the sample dataset (**lda_4800k_data.las**) and click Open. This will open the Data Files dialog.

** The Data Files dialog allows you to change the display Symbology of the lidar data in the Fusion window—but, I recommend that you accept the defaults for now*.

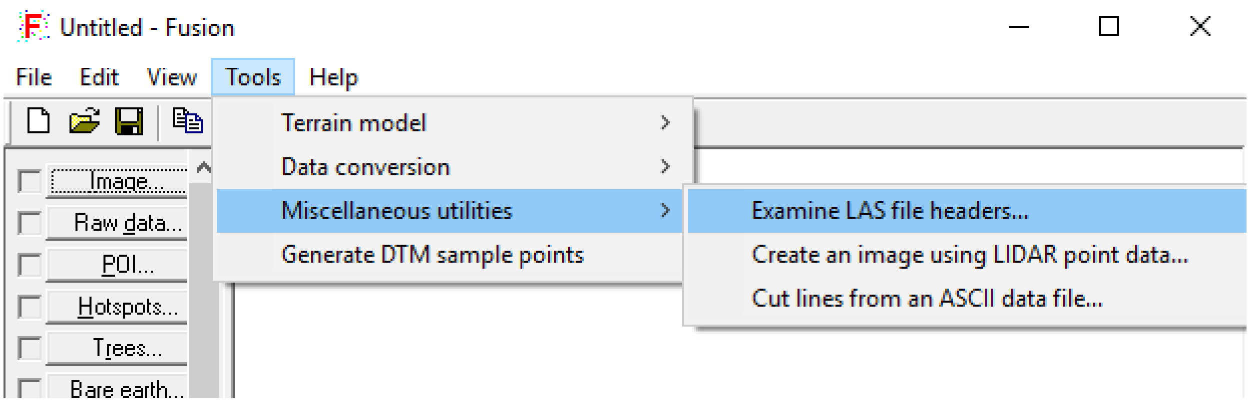
3. Click OK (read the note below before making any changes).

4. Save your Fusion project by clicking the Save icon or File > Save As.

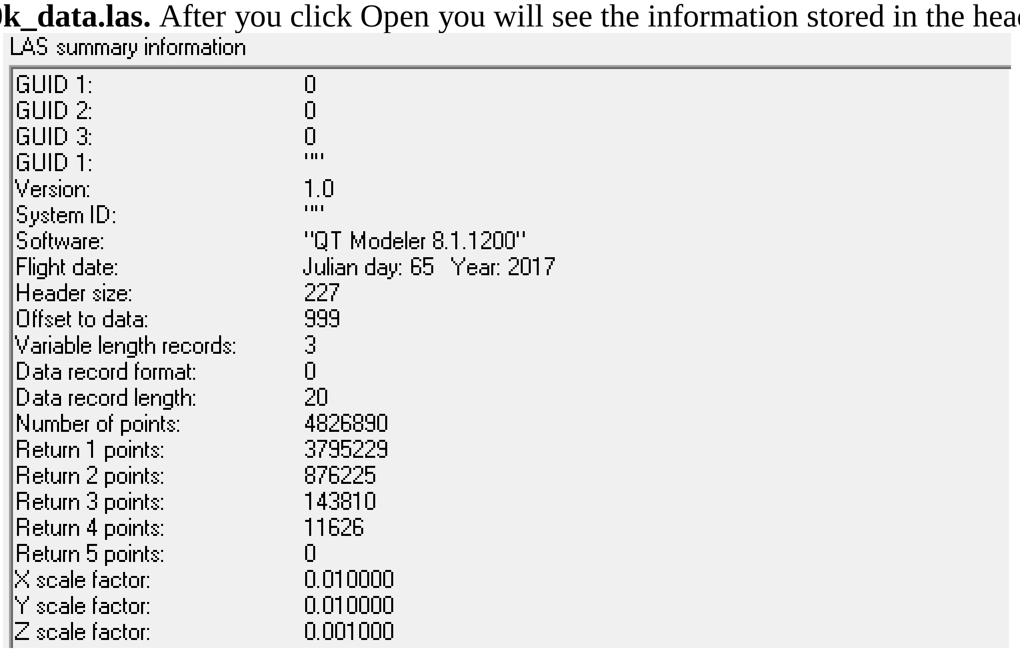
5. Name the project **exer1.dvz** and Save it somewhere easy to find.

B. Examine las header

Besides xyz coordinates of each return, a las file store information about the way how the point cloud was created, as well as some summary statistics, such as the number of points or min and maximum elevation. To see the las header, you should click Tools->Miscellaneous utilities->Examine las file header:



In the window that appears after you select Examine LAS headers, navigate to **lda_4800k_data.las.** After you click Open you will see the information stored in the header.



LAS summary information

GUID 1:	0
GUID 2:	0
GUID 3:	0
GUID 1:	""
Version:	1.0
System ID:	""
Software:	"QT Modeler 8.1.1200"
Flight date:	Julian day: 65 Year: 2017
Header size:	227
Offset to data:	999
Variable length records:	3
Data record format:	0
Data record length:	20
Number of points:	4826890
Return 1 points:	3795229
Return 2 points:	876225
Return 3 points:	143810
Return 4 points:	11626
Return 5 points:	0
X scale factor:	0.010000
Y scale factor:	0.010000
Z scale factor:	0.001000

9.5 Load a Reference Image

Click the Image button

1. Select the sample orthophoto (**orthophoto_4800k.jpg**) from your sample data folder and click Open. The image will automatically display in the Fusion viewer. Fusion requires a reference image before you can view your lidar data in Lidar Data Viewer (LDV). If you don't have a reference image available, you can create an intensity image from the raw lidar data to use as your reference image. This capability can be accessed through the FUSION interface: Tools > Miscellaneous Utilities > Create an image using LIDAR point data.

i. If you want to zoom-in to any part of the image, Right Click on the location you want to zoom to. Zoom-out by holding down shift and right-clicking the image.

ii. You're now ready to view the lidar data in the Lidar Data Viewer (LDV)

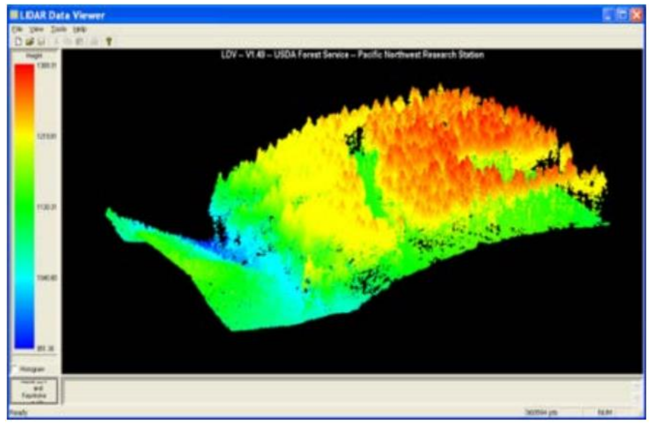
9.6 View a Sample in LDV

To create a sample and view the corresponding Lidar data in 3D:

1. Position the cursor over an area of interest in the orthophoto, left click and drag a small box (called a stroked box) over the area and release the left mouse button—this is your sample (see the following figure).

Note: a small sample works better (faster) than a large sample. If the area you selected is not a square, it means that the shape of selection is set to a fixed box, fixed circle, stroked circle, or corridor. To change it to a stroked box (basically a rectangle with side not preset) click on Sample Options, then change the shape of sampled points to Stroked box.

2. The sample box will be highlighted in the Fusion viewer and LDV; Fusion's 3D viewer will automatically appear and load the Lidar data within your sample boundary, similar to the following figure. The default coloring scheme is color the points based on height, in actuality elevation, but if you want to change it, you can select Sample options, then choose a different option, such as color by classification or intensity.

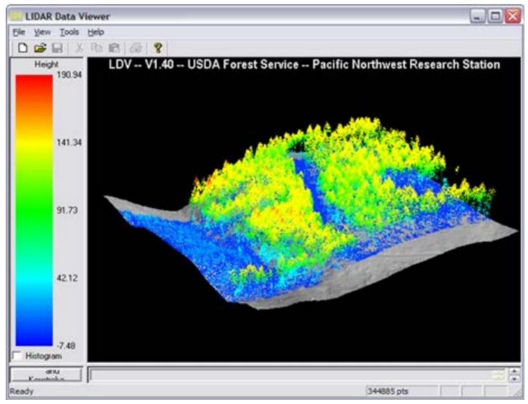


3. Use the Basic LDV Navigation Tips (see below) until you are comfortable with your ability to control the data cloud. The LDV contains two image areas: the point cloud, the largest image displayed in the LDV, and the summary statistics associated with the point cloud, which is the colored bar on the left. Depending on the selected attribute, the bar could show the height or intensity. In the image above, the height is displayed. The largest value is the largest elevation, which depends on whether or not the ground is subtracted from the elevation of each point. The process of subtracting the ground from each point is called normalization of the point cloud, and the product is a normalized point cloud. A normalized point cloud has the ground at 0, and if you look at it from the side it will have a flat bottom.
    a. Tips with Basic LDV (Note: LMB = Left Mouse Button, RMB = Right Mouse Button):
        i. LMB + move mouse: "Grab" and rotate the displayed data (rotation can be up/down and left/right)
        ii. It may be easiest to imagine that the data is contained in a glass ball. To rotate the data, use the mouse to roll the ball and thus manipulate the data.
        iii. LMB + ctrl + move mouse down: Zoom in
        iv. LMB + ctrl + move mouse up: Zoom out
        v. RMB: Activate the pop-up options menu for LDV
        vi. For a complete list of LDV keystrokes, Click the About LDV and Keystroke guide button in the bottom left corner of the LDV (highlighted in red in the graphic to the right).

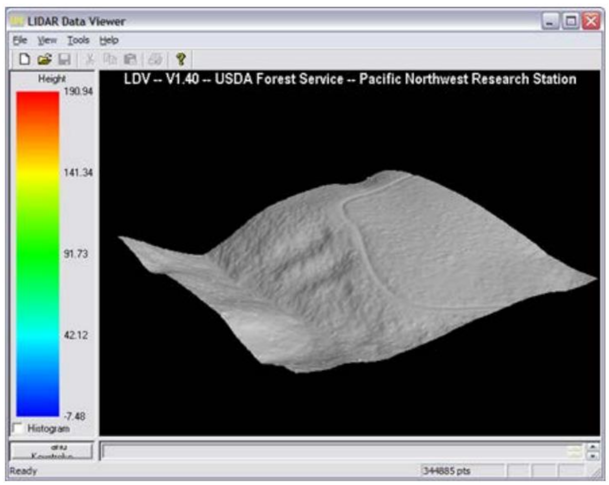
9.7 Add a Bare Earth Model

Close the LDV and return to FUSION (it will still be running & your last sample will be displayed).

1. Click the Bare earth button (located on the Fusion toolbar).
2. Select the terrain model (**4800K_ground_surface.dtm**) from the folder and click Open.
3. Within the Surface model options window, you can accept the defaults or define contour intervals and line colors. Once you have chosen intervals and colors or accepted the defaults, click OK.
    i. The terrain model will be displayed in the viewer as a contour map over the orthophoto.
4. Click the Repeat last sample button (located on the Fusion toolbar). This will display the same lidar data cloud as before.
5. Right-Click within the LDV viewer to access the "Right Click Menu".
6. Click on Surfaces (or use the Alt-U keyboard option). The bare earth surface will automatically display with your lidar data cloud (see the following figure).



7. Access the right-click menu again and Click on Data to toggle the data off (or type Alt-D). This will allow you to inspect the bare earth surface without the data cloud (see the following figure). To turn the data back on using the right-click menu and click on Data again or type Alt D again.



9.8 Explore Fusion's Sampling Options

A. Change sample options

1. In the FUSION window, Click the Sample options button to open the Sample Options dialog.

2. Under the Sample shape section, select Stroked circle (the default is Stroked box) and click OK.

3. Select a small stroked-circle sample in the Fusion window and view the results in the LDV.

4. Close the LDV.

B. Return to the Sample options

1. Change the sample shape back to a Stroked box.

2. In the Decimation section, increase the value to 200 and click OK.

3. In the Fusion window, select a large stroked-box sample (make the stroked box cover about half the size of the reference image).

i. It may take a few minutes to extract the sample, but the results display very fast in LDV. Notice that the ground is not flat within the sample area as you view the data in the LDV; you'll make it flat in the next sample.

4. Return to the Sample options and select (check) the Subtract ground elevations from each return in the Options section and click OK.

5. FUSION Tip - you can only use the Subtract ground elevations option if you've loaded a Bare Earth Surface (which you've done).

6. Click the Repeat Last Sample button. This will repeat the last sample area, but now the data will appear in LDV as flat terrain—this is very useful for comparing heights above ground level.

7. Return to the Sample options and change the Decimation value back to 1 and deselect the option to Subtract ground elevations.

8. Select the Bare Earth Filter option to Exclude points close to surface and increase the tolerance to 2.

9. Click OK to close the sample options dialog.

10. Select a small stroked box sample in the Fusion window

Note that the points close to the ground have been excluded from displaying in LDV.

11. Return to the Sample options and select the Include all points option under Bare Earth Filter, and select the color using image option under the color menu.

12. Click the Repeat last sample button. You should notice that each lidar return is now painted the color of the corresponding reference image. Keep the LDV open.

13. FUSION Tip - Some of the sampling and display options can be very computer-intensive and are appropriate only with a fast computer and a small sample area. You will have to be the judge of what your computer is capable of handling. A class 3 computer is recommended for general lidar data exploration and analysis.

9.9 Explore Fusions Display Options

A. Explore a few of the display options within LDV

1. Access the right-click menu and toggle (either on or off) the Draw all points when moving option. Click-and-drag to move the data sample. If you've toggled the Draw all points option on, the responsiveness of your display may be sluggish (but it looks good), and if you toggle the option off, the LDV display will be very responsive (but it won't look as good).

2. Set the Draw all points option to suit your computer and your preferences.

3. Access the right-click menu again and click the Marker option.

4. Experiment with the Marker Type and Marker Size options—however, be aware that some of the marker types only work well with fast computers. If in doubt, keep the marker type set to Points.

5. Back in the Fusion window, Click the Sample Options button.

6. Enable the Color by Intensity option.

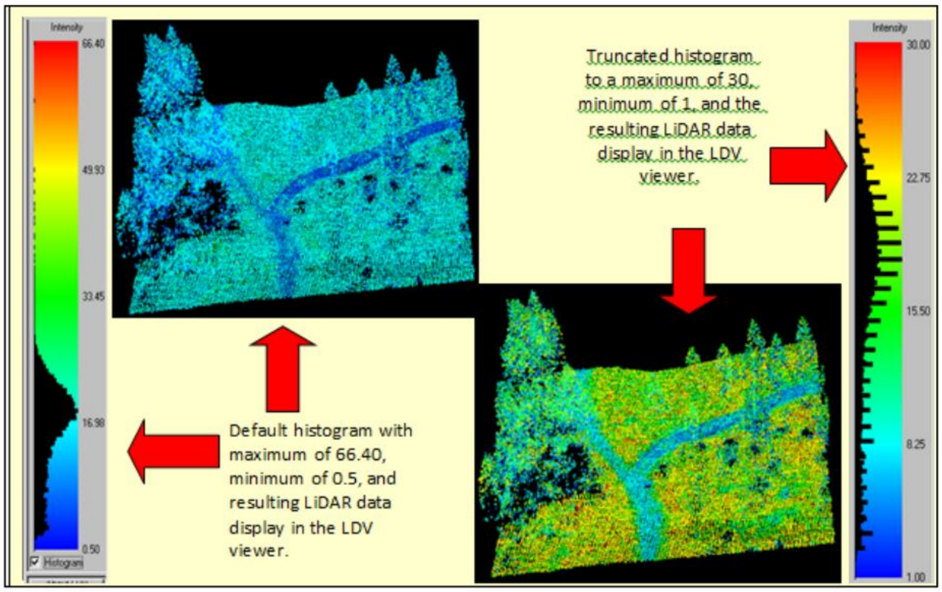
7. Click OK and then Click Repeat Last Sample.

i. The LDV viewer will display returns according to their intensity value (or the near-infrared spectral value). The intensity information can be helpful to interpret ground features. However, because the intensity information of the ground features are clustered on only a small portion of the displayed intensity range, the default display parameters make the data difficult to interpret. Let's adjust the intensity display parameters to improve interpretation.

B. Click the Histogram checkbox on the left side of the LDV viewer (see following graphic).

1. The histogram will display (black) along with the color legend
2. You should see that most of the intensity values are clustered in the lower half of the available intensity range. Let's truncate the available range to the approximate range of the intensity values (so that we can interpret the LiDAR data better).
3. Write down the approximate low and high-intensity values that capture most of the histogram.
4. Click the Sample Options button in the Fusion window.
5. Enable Truncate Attribute Range (in the Color section).
6. Enter your approximate minimum and maximum histogram values.
7. Click OK and then Click Repeat Last Sample.

ii. Now, you've effectively stretched your intensity data to cover the full-color legend for improved interpretation (see following graphic). High intensity values (or high near-infrared values) for natural communities most likely represent photosynthetically active vegetation, and in some cases, may represent dry bare soil. Lower intensity values likely represent: 1) wet, bare soil 2) water or 3) less photosynthetically active vegetation.



C. Continue to interact with the data in LDV and experiment with the following items on the right-click menu.

1. Wiggle-vision (Alt-W)
2. Overhead View (Alt-O)
3. Reset Orientation (Alt-R)
4. Reset Zoom (Alt-Z)
5. Image Plate (Alt-P)
6. Turn the truncate function off and choose the color by the option that seems most appropriate for general visualization (color by height is recommended).
7. Save the project: File > Save.

9.10 Extracting Plot Subsets

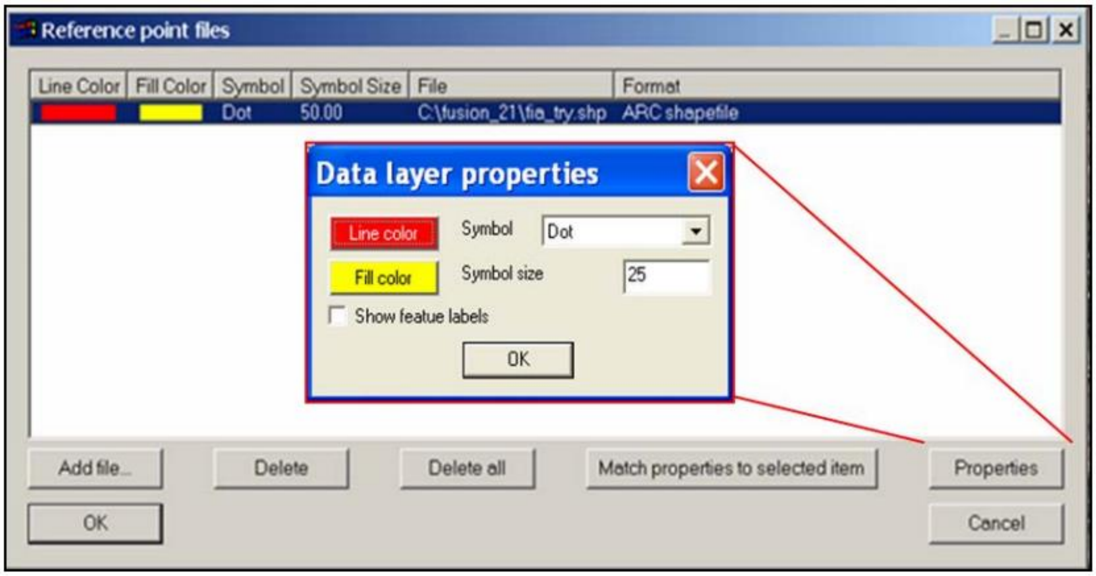
This exercise builds upon the previous exercise and uses the same sample data. You should have FUSION up and running with the **Orthophoto_4800k.jpg** image file displayed and the **lda_4800K_data.las** raw data file loaded (but not displayed). We have coordinates of the center points of three plots that have been converted to a shapefile (**three_plots.shp**).

9.10.1 Subsetting One to a Few Plots

A. Loading a shapefile as a POI (Point of Interest) in Fusion

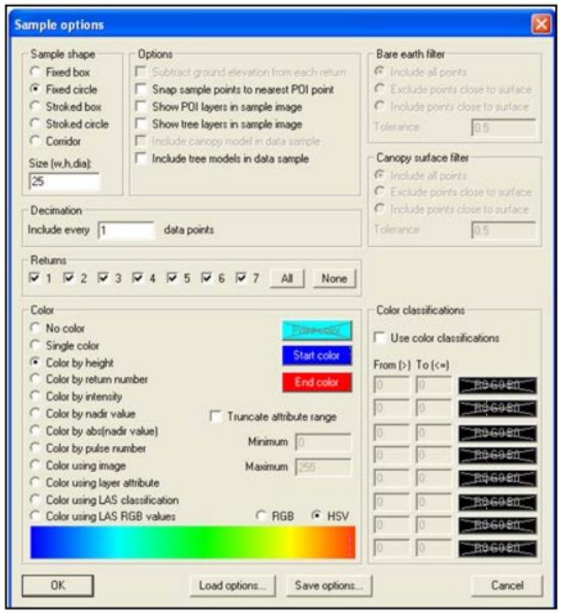
1. Click the POI button.
2. Navigate to the location of the three_plots.shp shapefile.
3. Select the file and click Open.
4. Change the attributes (color and size) of the POI if desired (see following graphic) and click OK. The three plot locations will display over the orthophoto.

** If you've already loaded the bare earth surface for this project area, skip to step 9.



5. Click the Bare earth… button.
6. Navigate to and select **4800K_ground_surface.dtm**.
7. Click Open.
8. Accept all of the surface model default options and click OK. Topo lines representing the ground surface elevations will display over the orthophoto image.
9. Now, we only want to select the lidar data within our plots rather than the entire lidar dataset.

B. Click the Sample options button (see following graphic) and select the following options



1. Sample shape: Fixed circle.

2. Size: 120 (diameter in feet) Note: in future projects, make sure the units of the data and the plot are the same. If the data are in UTM meters, convert the plot diameter to meters as well.

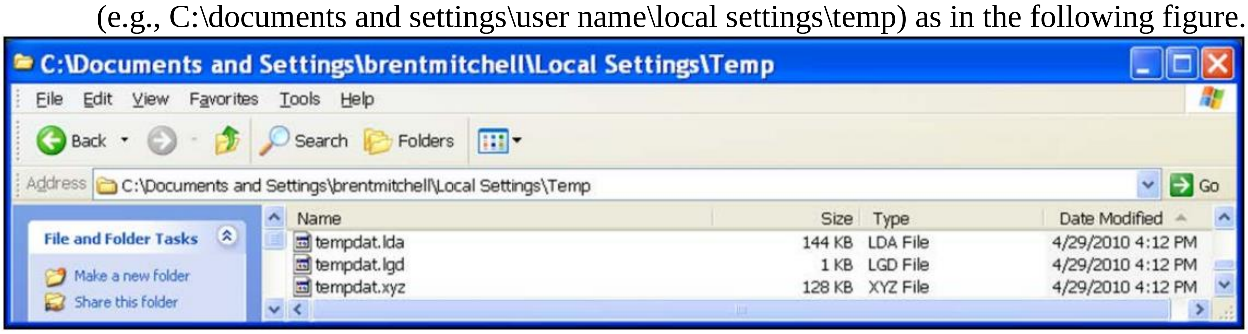
3. Options: activate (put a checkmark in the box) Snap sample points to nearest POI point.

4. Options: activate Subtract ground elevation from each return.

5. Click OK at the bottom left to accept the sample settings.

6. Next, within the Fusion display window, click on one of the plot locations (note which one you select). The POI changes color and an LDV window pop up showing the lidar points within the 120-meter diameter circle around the plot center.

i. These lidar data points are simultaneously written to three temporary files named tempdat with three different extensions (.lda, .lgd, .xyz) in the user's temporary folder (e.g., C:\documents and settings\user name\local settings\temp) as in the following figure.



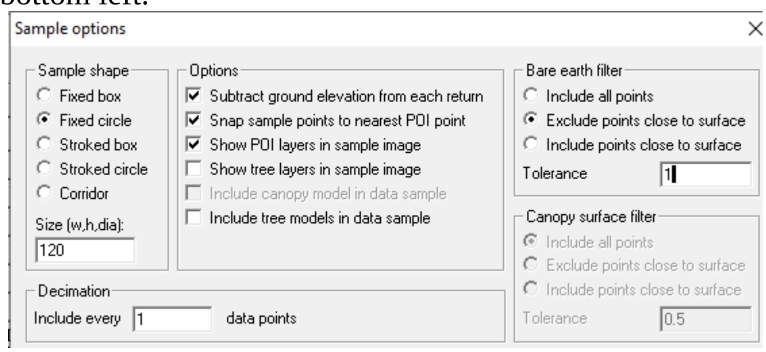
ii. Potentially, you could locate and rename the 3 temporary files with a corresponding plot identifier: ex Plot1.* and move them into a working directory. You would then have a subset (for each plot) of lidar data. It is good to be aware that FUSION writes these temp files and deletes them when FUSION closes (if you want the temp files, make sure you move and rename them!) In the next lab, we will work through a process that will allow you to automate the subsetting of many plots and subsequently calculate statistics for each plot.

9.11 Measurements of individual trees

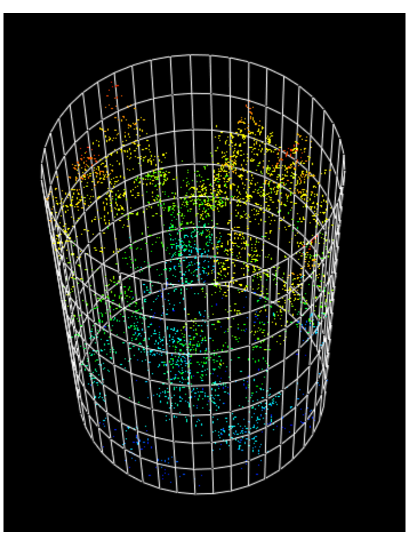
In many situations, you are asked to measure the height of individual trees. Fusion can execute this task easily, as it has a dedicated feature for manual measurements of tree height. To start this process first, you have to subset the data such that the focus will be only on a small number of points.

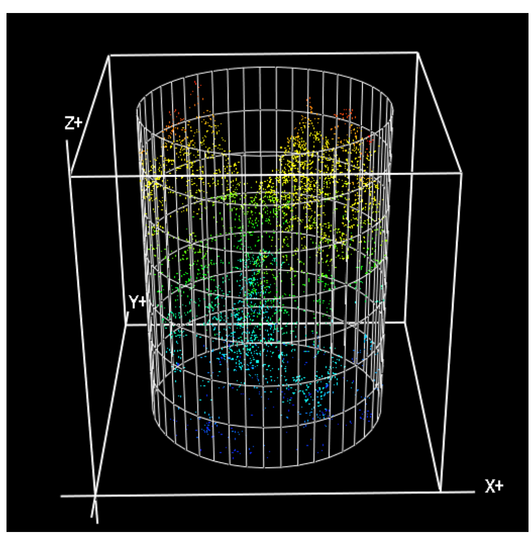
In the main Fusion window, click the Sample options button and select the following options:

1. Sample shape: Fixed circle.
2. Sample Size: 120 (diameter) Note: make sure the units of the data and the plot are the same, if the data are in UTM meters, convert the plot diameter to meters as well.
3. Options: Subtract ground elevation from each return.
4. Options: Snap sample points to nearest POI point.
5. Options: Show POI layers in sample image.
6. Bare earth filter: Exclude points close to the surface.
7. Bare earth filter: Tolerance 1 (the tolerance is the distance from the surface in the same units used for LIDAR data elevations).
8. Click OK at the bottom left.



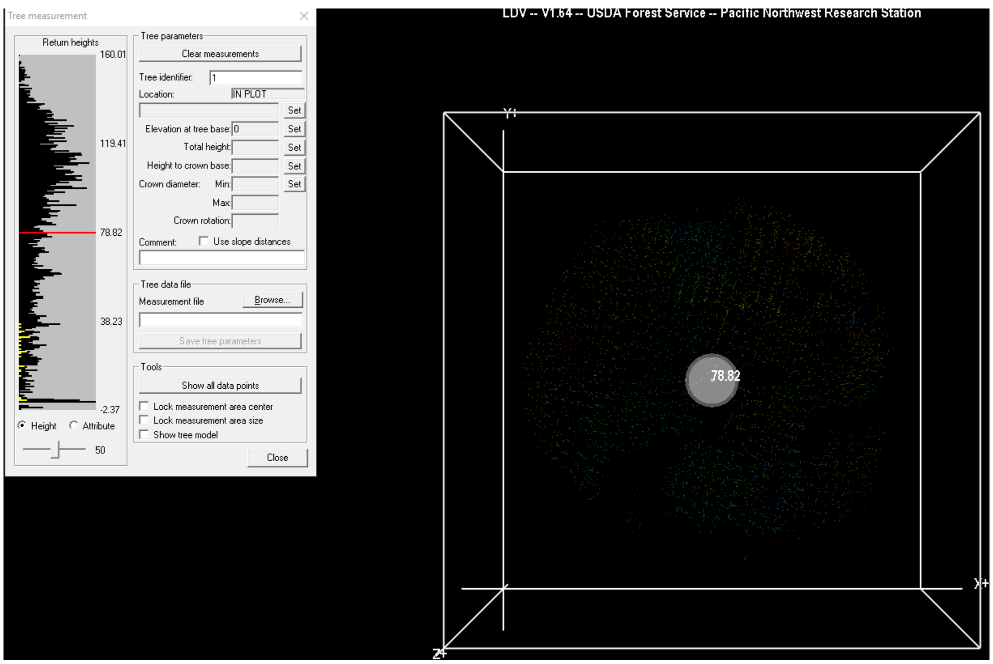
9. Toggle the checkmark for Plot mode on.
10. Toggle the checkmark for Display sample on.
11. Then click on the location of the plot from which tree information will be extracted (the lower-right most of the three plots is the easiest of the plots to use). The subset of data will pop up in the LDV window (see following graphic).





Note: if the axis and plot cylinder do not show, right-click in the LDV window to display the LDV right-click menu. Click on Colors menu item and set the axis color to a different color than the background color, let say white. If the axis shows but the plot cylinder does not, then ensure that the Plot mode button is checked in the Fusion window.
You should notice that the height of the points is between a small negative value and less than 200 (the colored bar on the left), which indicates that the values represent the actual height of the points above ground rather than elevation. The presence of negative values for height is due to the ground interpolation algorithm, which creates a surface best adjusted to the data. Therefore, some points could be above, some below, and some on the surface. Also, most of the ground points are excluded from the data set. If you wish to include them in the sample display, return to the Sample options and select the Include all points under the Bare earth filter options, and then click the Repeat last sample button. However, when you are doing individual tree measurements, it is advisable not to display the ground points because the focus is on the top part of the crown.

To measure individual trees, Fusion has a special tool called Trea Measurement. To turn on the Trea Measurement window, you should either press F9. The Tree Measurement tool allows us to isolate a single tree within the plot and make measurements for that tree and more. The Measurement marker will display automatically—however, you can right-click anywhere in the LDV window to bring up the right-click menu and click on the Measurement marker to toggle the Measurement marker on. A cylinder will show in the LDV window.



Note: The yellow histogram represents the lidar points within the measurement cylinder; the black histogram represents all of the lidar points within the data subset (in this case, the circular plot). The yellow histogram can be proportionally scaled relative to the entire plot histogram by sliding the control below the histogram.
12. Center the cylinder over a tree within the plot:
i. This cylinder can be moved by using the combination of ctrl + right mouse + movements. **Tip**: once you have the cylinder centered over a tree, it's a good idea to click the Lock measurement

area center checkbox on (remember to unlock it when you want to move the cylinder to another tree).
ii. The diameter can be changed by shift-ctrl-right mouse.
iii. The shape or aspect ratio can be changed from circle to ellipse by using ctrl and up or down arrows.
iv. The orientation of the ellipse is modified with ctrl + left and right arrows.

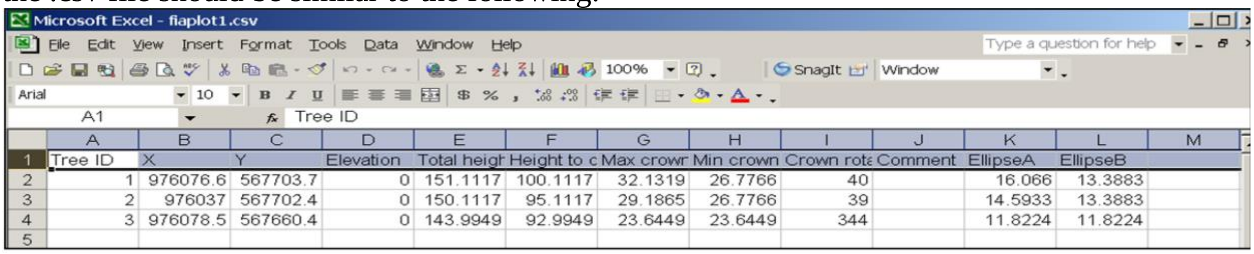
Note: Using these key and mouse combinations is not immediately intuitive (to put it mildly). However, it is important to fit the 3D measurement cylinder as closely as possible to the 3D shape of the tree.

v. As you change the position, shape, and size of the cylinder, note how the histogram changes in the Tree Measurement window.
vii. After the cylinder is centered over a tree, measurements can be made from the data using the red lever manipulated by the mouse wheel. Some keystrokes to snap the marker to the top or bottom are available too. The list of keystrokes can be accessed by clicking on the lower-left button of the LDV window.

To use the Tree Measurement tool to record some tree measurements, you should follow a series of steps:

1. In the Tree Measurement tool: Add an entry for a Tree identifier (the default of 1 is sufficient since this is the first tree of this plot).
2. Location: set the location of the cylinder center by clicking the adjacent Set button.
3. Elevation at tree base: press “l” (lower-case L) to drop the marker to the lowest point in the cylinder (don’t press upper-case L or you’ll move the cylinder) and then scroll the Measurement marker up/down to the tree base (check that this is set to zero or a number slightly above zero else tree models that you will soon create won’t display) and click the adjacent Set button.
4. Total height: press “h” to raise the marker to the highest point in the cylinder (don’t press “H” or you’ll move the cylinder) and scroll the Measurement marker to the top of the tree and click the adjacent Set button.
5. Height to crown base: scroll the Measurement marker down to the crown base and click the adjacent Set button.
6. Tip: try moving the measurement plate to the base of a crown (up/down), then make sure the cylinder is larger than the crown and press the “F” key. This fits the cylinder to the crown points.
7. Crown diameter: min, max, and crown rotation: These three values are extracted from the cylinder diameter measurements when you click the adjacent Set button (Note: if min and max are the same value, you’ve used a simple circular tree crown model (not an ellipse). If the crown rotation angle is zero, you’ve not rotated the ellipse.).
8. Comment: add an optional comment if you notice something that is special about tree, such as broken top, or curvature of the stem.
9. Create a new .csv file or select an existing .csv file to save the measurements by clicking on the Browse button.
10. Click Save tree parameters. Note: After the parameters are saved, the points belonging to the just measured tree disappear from the screen (they can be brought back by hitting the show all data points but they cannot be turned off again. Therefore, while you are in the measurement mode, you should refrain from bringing back the points of measured trees, as it will hinder the measurements). The tree identifier increments automatically.
11. Uncheck the Lock measurement area center checkbox.

12. Move the cylinder to the next tree and check the Lock measurement area center checkbox. 13. Adjust the size, shape, and orientation of the cylinder to fit this next crown
14. Set all of your measurements in the Tree Measurement dialog for this tree.
15. Click Save tree parameters. The measurements and comments (if any) will be appended to the previous .csv file.
16. Repeat these last five steps (beginning with uncheck the Lock measurement area center…) for each tree in the plot.
17. Once done, open your recently created .csv file. For a plot with only three trees, the content of the .csv file should be similar to the following:



	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Tree ID	X	Y	Elevation	Total heigh	Height to c	Max crowr	Min crown	Crown rota	Comment	EllipseA	EllipseB	
2	1	976076.6	567703.7	0	151.1117	100.1117	32.1319	26.7766	40		16.066	13.3883	
3	2	976037	567702.4	0	150.1117	95.1117	29.1865	26.7766	39		14.5933	13.3883	
4	3	976078.5	567660.4	0	143.9949	92.9949	23.6449	23.6449	344		11.8224	11.8224	
5													

Questions

Submit a Word document that answers the following questions (each worth 10%):

Q1. How many points are stored in the lda_4800k_data.las file?
Q2. How many return points are in the lda_4800k_data.las file?
Q3. What is the coordinate system on which the points were stored in the lda_4800k_data.las file? You should know that the image and the points cloud have the same coordinate system, and you could use ArcGIS to find the information needed.
Q4. What is the maximum tree height on each plot?
Q5. What is the maximum and minimum elevation on each plot?
Q6. What is the maximum intensity value for the entire las file? There is a chance that rendering process will take some time, have patience. It is important to have the data stored locally.
Q7. Include a snapshot with the points from LDV before and after the ground elevation was subtracted from each return.
Q8. How many trees are in each plot?
Q9. What is the height of the second largest tree in each plot?
Q10. Why there are points with negative values after normalization of the point cloud.

10 Introduction to Lidar: Command line

Lab developed by Dr. Strimbu from USFS documentation.

10.1 Objectives

- Working with shapefile in Fusion
- Extract points from las file using a shapefile
- Normalize a point cloud
- Estimate tree metrics

10.2 Files used in this Lab

The files needed to complete this Lab are supplied by the U.S. Forest Service, and are on Canvas. Copy the files localy; such an address could be T:\Teach\Temp\JaneDoe\FE444\Lab10, where Jane Doe should be replaced with your name. This path would be used for all the subsequent examples, but you should customize to your settings.

Files (dat and hdr format)	**Description**
lda_4800k_data.las	Point cloud
Orthophoto_4800k.jpg	Ortho-rectified image
Orthophoto_4800k.jpgw	Ortho-rectified image georeferenced
4800k_DSM	Digital surface model
4800k_ground_surface	Digital terrain model
three_plots.shp	Shapefile for plots

10.3 Lab Procedure

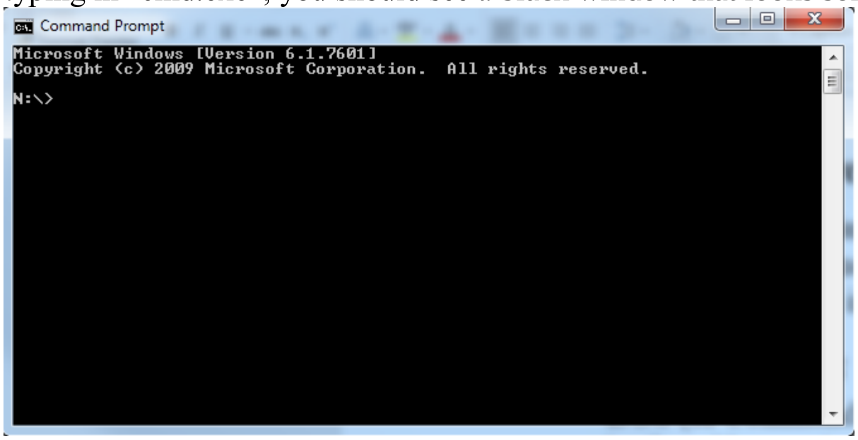
By now, you should be familiar with the basic features of Fusion. As with all new programs, it is helpful to play around and get a feel for the program. Today we will be diving further into the weeds of command-line interfacing with Fusion.

10.3.1 Files in Windows

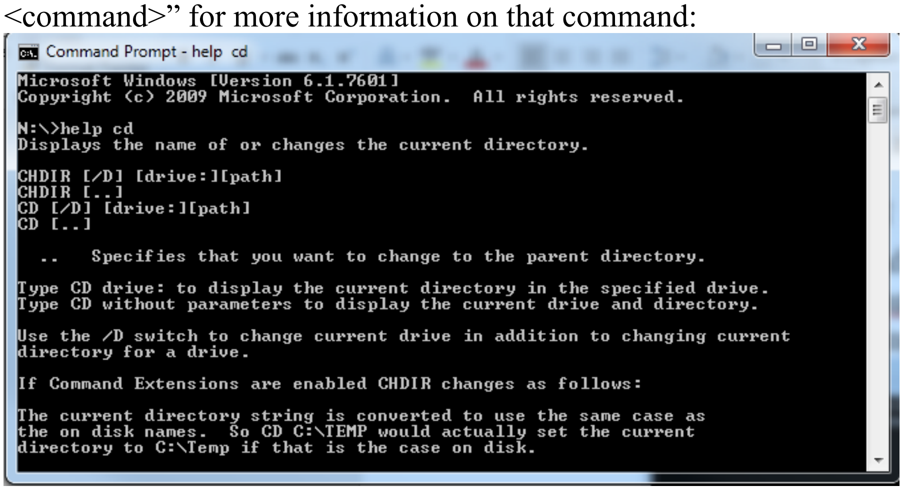
If you are not familiar with Windows environment you should know that there are two types of files: some that perform a task, which are called executable, and files that store information. The information can be defined by the used or can be accessed by the executable files, as they store critical pieces of data needed for correct running of the task. The executable files are identified with the extension exe while the batch files with the extension bat. By default a file is executed if it is enter at the prompt, as we will see later, or by double clicking it, in Windows explorer. In fact any time when you start a program an executable file is triggered. If only the name of the executable file is entered at the command line, then the file have to be located in the current directory listed by the prompt. If the executable file is located in a different folder then either the path to that location is typed or the location of the prompt is changed to the respective folder.

10.4 DOS

DOS, which stands for Disk Operating System, (or Command Prompt in Windows) is a command driven computing language. It can be found by opening the START menu and typing in "cmd.exe"; you should see a black window that looks something like this:



The directory you start in may change from computer to computer; the example above has us in the "N:\" directory. Type in 'help' to see a list of available commands, or "help <command>" for more information on that command:



To navigate the computer folder structure using DOS you have to use a particular syntax. First, it is recommended to change to the appropriate drive, if by default DOS is not pointing the drive that you want. In the example from above, DOS points to the N drive, whereas all Fusion tools are on C drive. To change from N to C you simply type **c:** or **cd c:**. Once on the appropriate drive you should go to the Fusion folder by typing either **cd c:\Fusion**. The **cd** DOS command stands for Change Directory.

Subsetting the LIDAR points for a substantial amount of plots can best be done from the DOS command line using FUSION's command-line executable **clipdata.exe**, which can be run on one plot at a time or on multiple plots by using a batch process. One method for doing this is by using MS Excel to create large batch files. The advantage of creating such a file in a spreadsheet is the capability to generate a series of incremented field entries and to be able to concatenate several fields into a single text string (this technique can save you a lot of time if you are trying to analyze hundreds of plots!). Please refer to ***Appendix 1*** (at the end of this lab) if you are interested in learning more about creating batch files in a spreadsheet environment.

Before you go on to run the clip data command, it's critical that you have a sound understanding of why and how to use the DOS command prompt and batch files to run Fusion utilities. If you aren't proficient with the basic skills needed to work in the DOS environment, you won't be able to efficiently handle Fusion projects. The following are some important facts and tips for using the FUSION commands:

- The FUSION command line utilities and processing programs are collectively referred to as the FUSION LIDAR Toolkit or FUSION-LTK.
- First of all, you must know which Fusion utilities will give you the most useful information for your project. To study Fusion utilities, click the Fusion Manual shortcut. Page 16 provides an introduction to using command line utilities and pages 17-20 give an overview of all Fusion utilities and processing programs. **The more you understand the capabilities of the FUSION commands, the more efficient you will be at extracting useful information from lidar data**. You will need to understand each command line utility enough to customize it for your data by manipulating the parameters and switches. The defaults will usually produce poor results!
- The FUSION-LTK programs are designed to run from a command prompt or using batch files. There is no friendly GUI available!
- Remember to follow these three guidelines as you work in DOS:
  - Good file organization– make sure your folders are organized and named in a simple and intuitive manner.
  - Correct Syntax– take your time and check before you run. ***Incorrect syntax can overwrite and delete your input file***!!!
  - Directory navigation- call the correct directory. A good file structure and proper syntax will make this much easier.
- Most of FUSION's command line *.exe files can be batched to facilitate/automate repetitive data processing steps from the DOS prompt. A batch file (see graphic below) is simply a text file containing a separate line for each DOS command or each time the command should be executed. These files can be created with a text editor or any other program that can save the output as a text file (i.e. Notepad or Excel). They are easy to edit and improve efficiency for large projects.
- The extension of the batch file has to be .bat to let DOS know it has to process every line consecutively. Running the batch file is easy--simply navigate to the directory containing the batch file in the command prompt and type the name of the batch file.

Recommendations on working with Fusion LTK

As I said, to run a task in Fusion you have to enter the name of the file executing the task at the command prompt or double clicking in Windows explorer, if no parameters are needed. However, Fusion LTK requires parameters which suggest that only the command prompt should be used. Typing on DOS is not easy, pleasant, and is prone to errors. Furthermore, once you submit the command, if wrong, you have to type it again, which in many cases is really frustrating. A work around this issue is either usage of text files or the creation of a batch file. In many instances you could run only one LTK command in batch mode, just because is easy to debug, if needed. Remember that a batch file, also called bat file, is an ASCII file that contains the succession of operations to be executed and is submitted to the CPU with the extension bat. We will see later how to use the bat files.

3.2 Explore the Clipdata command

1. Let's start by initially exploring the Clipdata command.
2. For the complete syntax, click the DOS prompt shortcut and type in Clipdata. However, if you are in a folder that does not contain the clipdata exe file then the command will not work. Therefore, either navigate to the folder where the clipdata command is located, usually in C:\Fusion or type the entire path: c:\fusion\clipdata.
3. Refer to the Clipdata section of the Fusion manual for more info.

DOS tip: You can use the up and down arrows to recall previous commands executed in the same command prompt window. This is very useful when you mess up the syntax and only need to change a character or two. You can edit the commands by moving the cursor with left and right arrows and using the insert key to control insert/overstrike

3.3 Example syntax

If you are in the Fusion folder then you can clip the lidar data by submitting the following command:
clipdata /shape:1 c:\lidar\sampledata\lda_4800K_data.las T:\Teach\Temp\JaneDoe\FE444\Lab10\clipplot1.las 975984 567628 976184 567828.
However, if you are not in the Fusion folder to run the same command you have to type the following command (considering that you do not want to navigate to the Fusion folder in DOS:
C:\Fusion\clipdata /shape:1 T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800K_data.las T:\Teach\Temp\JaneDoe\FE444\Lab10\clipplot1.las 975984 567628 976184 567828.
If the Fusion folder is installed under the folder **Program files**, or any other folder that contains spaces, then the syntax has to be changed to
"C:\Program Files\Fusion\clipdata" /shape:1 T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800K_data.las T:\Teach\Temp\JaneDoe\FE444\Lab10\clipplot1.las 975984 567628 976184 567828
As you see the path now is within quotation marks because a space means something for DOS, which means that you should specifically state that the space is a part of a path not part of the DOS command. This task is completed by including within quotation marks the path.
The parameters, or the switch as they are called in Fusion, required by the clipdata command and present in the example above are the following:

- clipdata is the FUSION executable command
- /shape:1 denotes a circular shape.
- T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800K_data.las specifies the input las file.
- T:\Teach\Temp\JaneDoe\FE444\Lab10\clipplot1.las defines the output (sample) las file.
- The last four numbers are geo-coordinates that define the bounding box of the circular shape.

If you want to have the point cloud normalized, meaning subtract the ground from the elevation of each return then you could enhance your command by adding the following switches: **/dtm:T:\Teach\Temp\JaneDoe\FE444\Lab10\4800k_ground_surface.dtm /height.** The final command should look like:

C:\Fusion\clipdata /shape:1 /dtm:T:\Teach\Temp\JaneDoe\FE444\Lab10\4800k_ground_surface.dtm /height T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800K_data.las T:\Teach\Temp\JaneDoe\FE444\Lab10\clipplot1.las 975984 567628 976184 567828

- clipdata is the FUSION executable command
- /shape:1 denotes a circular shape.
- /dtm:T:\Teach\Temp\JaneDoe\FE444\Lab10\4800k_ground_surface.dtm denotes the bare-earth surface model used to normalize the LIDAR data (subtract the bare-earth surface elevation from each lidar point elevation).
- /height is used in conjunction with the specified dtm to convert all elevation values to height above ground.
- T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800K_data.las specifies the input las file.
- T:\Teach\Temp\JaneDoe\FE444\Lab10\clipplot1.las defines the output (sample) las file.
- The last four numbers are geo-coordinates that define the bounding box of the circular shape.

After submission and complete execution of clipdata command, navigate to the output folder directory (T:\Teach\Temp\JaneDoe\FE444\Lab10) and verify that the files **clipplot1.las** was created.

REMEMBER: make sure that you use the proper slashes. *Forward slashes* (/) for switches and *back slashes* (\ ) for file paths. **Bad things can happen if you use the wrong slash!** Also remember to place a single space between all commands, switches, and file paths.

10.4.1 clipplots batch file in 'Notepad'

You can create the three las files with points for each plot by typing the command on DOS three times and changing the coordinates. However, you can do the same task in Notepad, which is way more user friendly than DOS, then submit the all the code at once as a batch file.

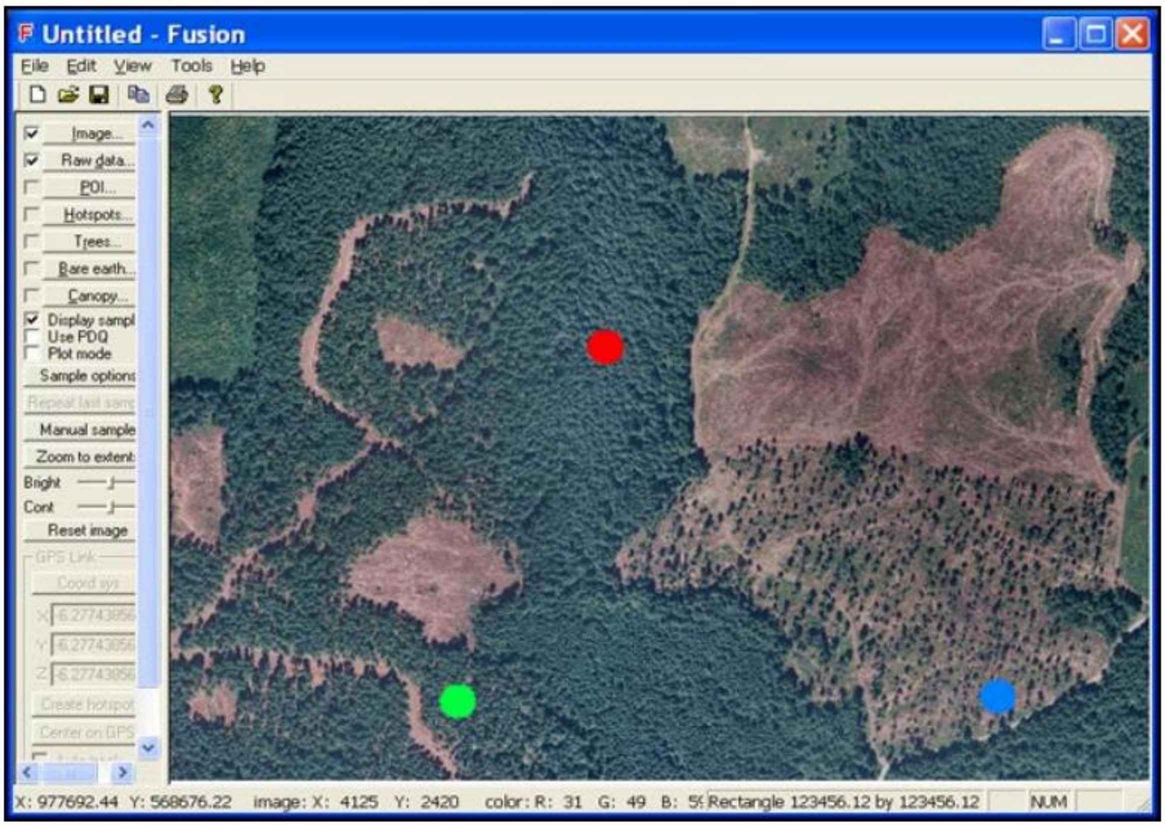
To create a batch file simply copy the syntax for plot 1 three times, then change the coordinates. The syntax should look similar to the following code where 3 plots are clipped and normalized (the color shows the there is one line of code for each plot, even that it is on two lines in Word):

```
c:\Fusion\clipdata /shape:1 /dtm:T:\Teach\Temp\JaneDoe\FE444\Lab10\4800k_ground_surface.dtm /height
T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800K_data.las T:\Teach\Temp\JaneDoe\FE444\Lab10\clipplot1.las 975984 567628 976184 567828
c:\Fusion\clipdata /shape:1 /dtm:T:\Teach\Temp\JaneDoe\FE444\Lab10\4800k_ground_surface.dtm /height
T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800K_data.las T:\Teach\Temp\JaneDoe\FE444\Lab10\clipplot1.las 977110 566582 977230 566701
c:\Fusion\clipdata /shape:1 /dtm:T:\Teach\Temp\JaneDoe\FE444\Lab10\4800k_ground_surface.dtm /height
T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800K_data.las T:\Teach\Temp\JaneDoe\FE444\Lab10\clipplot1.las 974760 566560 974880 566680
```

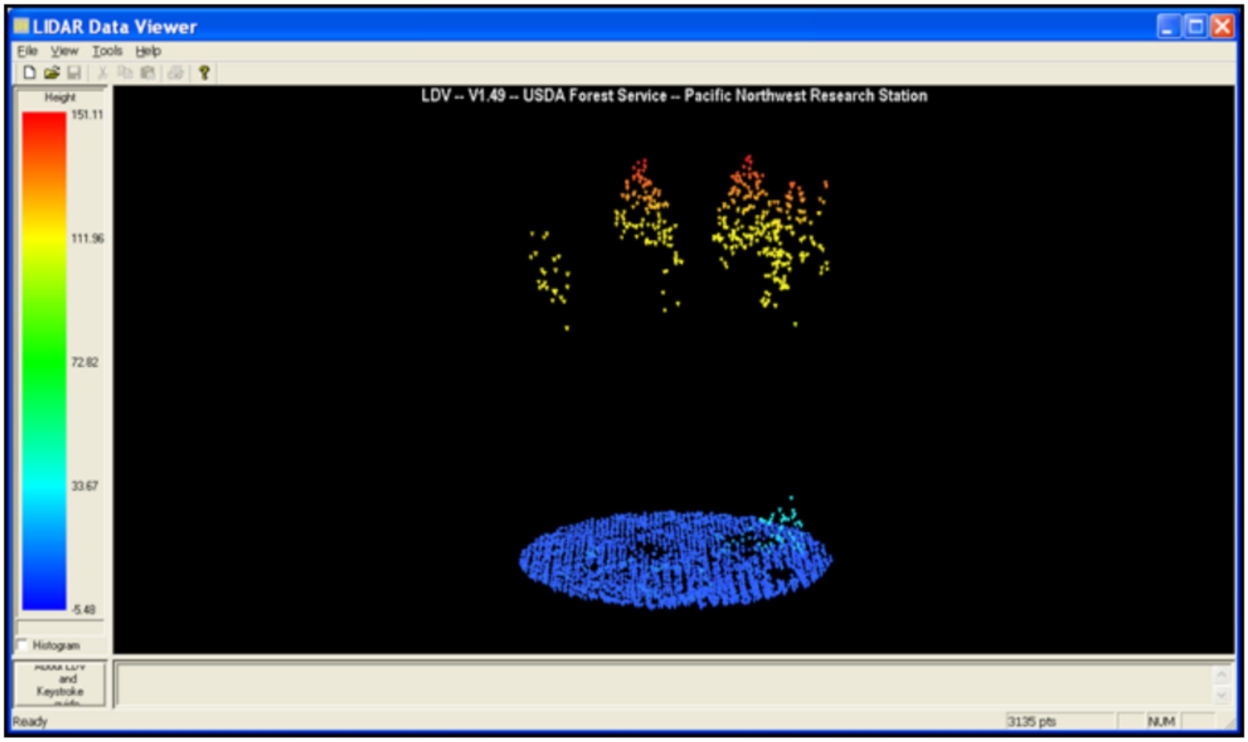
Tip: An easy way to avoid typos when entering file paths is to navigate to the file in windows explorer and then copy and paste the path into your batch file from the address line

10.4.2 View plot data in Fusion

1. Ensure that the Image File (**Orthophoto_4800k.jpg**) and that the three POI's from Part 1 of this exercise are loaded and visible.
2. Click the **Raw data** button in the Fusion window.
3. If the file **lda_4800K_data.las** is still loaded, click the Delete button to remove it.
4. Click the **Add File** button.
5. Navigate to and select **clipplot1.las**.
6. Hold the shift key down and select the last file, **clipplot3.las**, to select all three lidar plot subset files.
7. Click the **Open** button.
8. Give each data file a different color and change the **Symbology**. Select one of the rows (each row is a data file) in the Data Files dialog and for each selected row:
    a. Click the **Properties** button (or double-click the item).
    b. Change the **Symbol** to **Single Pixel**.
    c. Change the **line color** to a distinct color.
    d. Click **OK**.
9. Click the check box next to the **Raw data** button to display the three lidar subsets. Your results should be similar to the following graphic.



10. In your **Sample options** dialog ensure that you have selected a **Stroked box** sample shape and click OK.
11. Now, drag a sample over one of the plots and the lidar subset will display in the LDV, as in the following graphic.



10.5 Plot Metrics

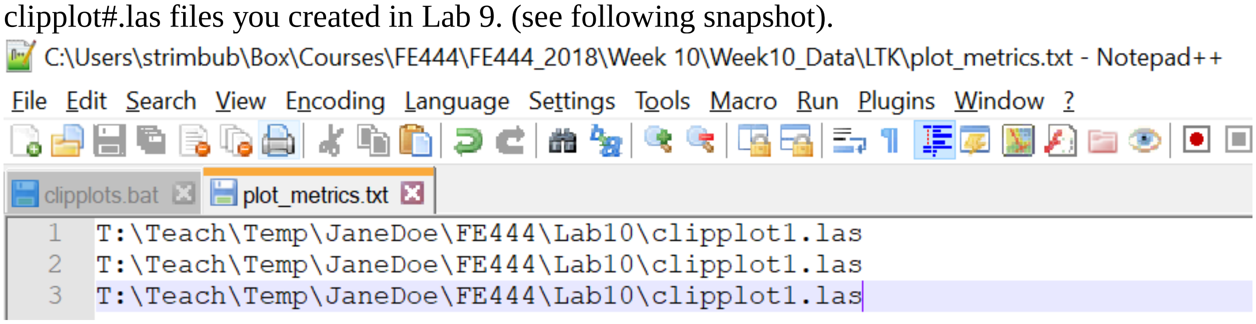
Various plots metrics can be computed with LTK. By now you should know that typing a LTK command anywhere on DOS will not work, except if you are located in the proper folder, which as I said, is likely to be C:\Fusion. To extract the cloud metrics using the command prompt first open the DOS Command prompt, navigate to c:\Fusion folder, and type **cloudmetrics** to explore the syntax of the cloudmetrics command.

Navigate to the **CloudMetrics** section in the Fusion manual for more info.

1. The syntax for the cloudmetrics command is:

   CloudMetrics [switches] InputDataSpecifier OutputFileName

2. The **InputDataSpecifier** can be a LIDAR data file, name of a text file containing a list of lidar file names (must have .txt extension), a catalog file, or individual file name. In our case, we will use a text file containing a list of lidar file names. Let's create that now.
3. Use Notepad to create the **InputDataSpecifier**: a text file containing the list of the three clipplot#.las files you created in Lab 9. (see following snapshot).

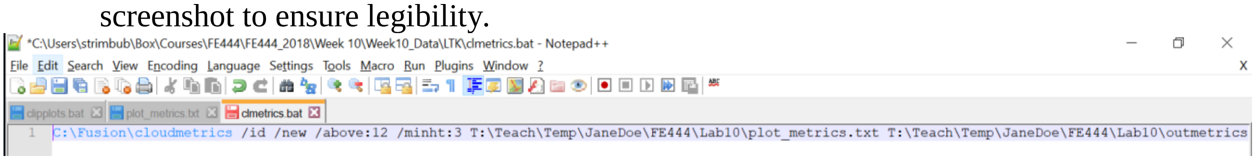


* They may also be called "**Plot1.las**" "**Plot2.las**" and "**Plot3.las**"

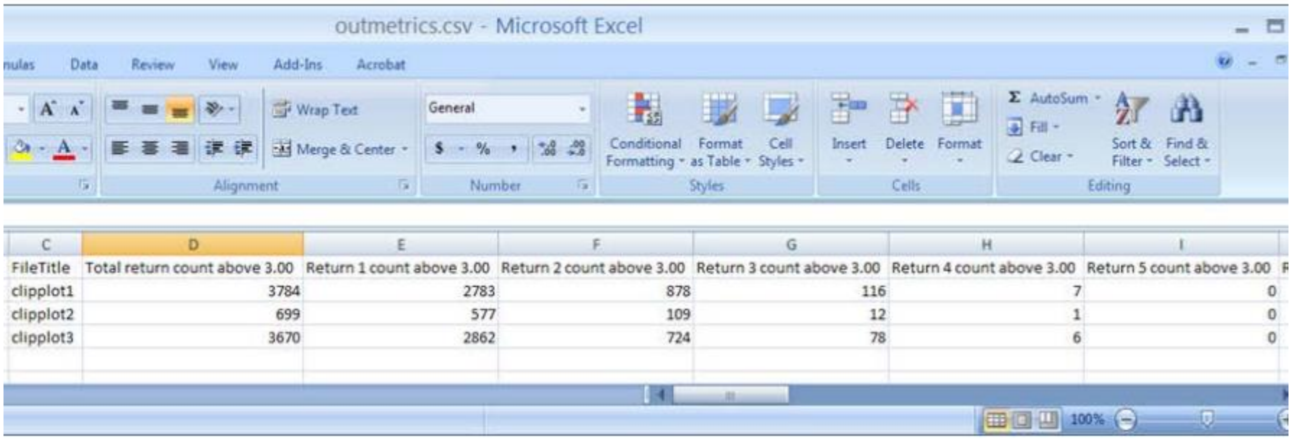
4. Name the file **plot_metrics.txt** and save it in the **T:\Teach\Temp\JaneDoe\FE444\Lab10** folder.
5. Open **Notepad** to create a **cloudmetrics** batch file. Use the following switches and parameters to create your batch file:
    a. **C:\Fusion\cloudmetrics** Fusion's command line executable
    b. **/id**, use file name to identify output
    c. **/new**, creates a new output file and deletes any existing file with the same name.
    d. **/above:12**, compute various cover estimates using the specified height break, in this case the height break is 12 feet. Please refer to Appendix 1 of this chapter for how cover metrics are computed.
    e. **/minht:3**, only use returns above a certain height to compute metrics. By choosing 3 feet we will exclude "ground" returns from our output stats. The units are in feet because the las file has the coordinates x and y in m and the elevation in feet. Fusion does not know how to convert from one coordinate system to another, so you should always be aware that all input files should be in the same coordinate system.
    f. **T:\Teach\Temp\JaneDoe\FE444\Lab10\plot_metrics.txt** InputDataSpecifier, in this case we have used a text file containing a list of the multiple file names, with full file paths included.
    g. **T:\Teach\Temp\JaneDoe\FE444\Lab10\outmetrics.csv** Output Data File, this designates the output file name. Note: we include the full file path, if the path is incorrect, or doesn't exist, the output file will not be created!

Your syntax should look similar to the following:
C:\Fusion\cloudmetrics /id /new /above:12 /minht:3
T:\Teach\Temp\JaneDoe\FE444\Lab10\plot_metrics.txt
T:\Teach\Temp\JaneDoe\FE444\Lab10\outmetrics.csv

6. Once completed save the file as **clmetrics.bat**. The csv part is missing from the screenshot to ensure legibility.



7. In DOS at the **T:\Teach\Temp\JaneDoe\FE444\Lab10** prompt, type **clmetrics** and hit **Enter**. Conversely, navigate to the location of the clmetrics.bat file and double click it.
    a. If you receive an error or the command will not run (after you have checked your syntax, please refer to Appendix 1 of this document).
8. After the program finishes, open the output file (outmetrics.csv) in Excel. The output .csv file looks close to the follows screen shot (one output line for each of the input data files):



FileTitle	Total return count above 3.00	Return 1 count above 3.00	Return 2 count above 3.00	Return 3 count above 3.00	Return 4 count above 3.00	Return 5 count above 3.00
clipplot1	3784	2783	878	116	7	0
clipplot2	699	577	109	12	1	0
clipplot3	3670	2862	724	78	6	0

Not only does FUSION calculate metrics for elevation of each return, it also calculates the metrics for intensity if the information is available. At this point in the development of aerial discrete return lidar technology, the intensity values are not normalized, so they are not ideally suited for analytical work. Hopefully in the future this will change and these metrics can be used in more analysis

Lidar has proven itself, through validated research, to directly and accurately measure height and % canopy cover of forest vegetation. So let's now take a closer look at our plots to see if we can understand how the metrics reflect what we see in the field and the lidar point cloud.

We have presented a variety of information for each plot in the following figure. Our hope is that by presenting the lidar point cloud visualizations, a site photo, and a selection of the cloudmetrics all together you will start to understand how lidar technology measures/collects information about forest landscapes. Let's point out some obvious correlations:

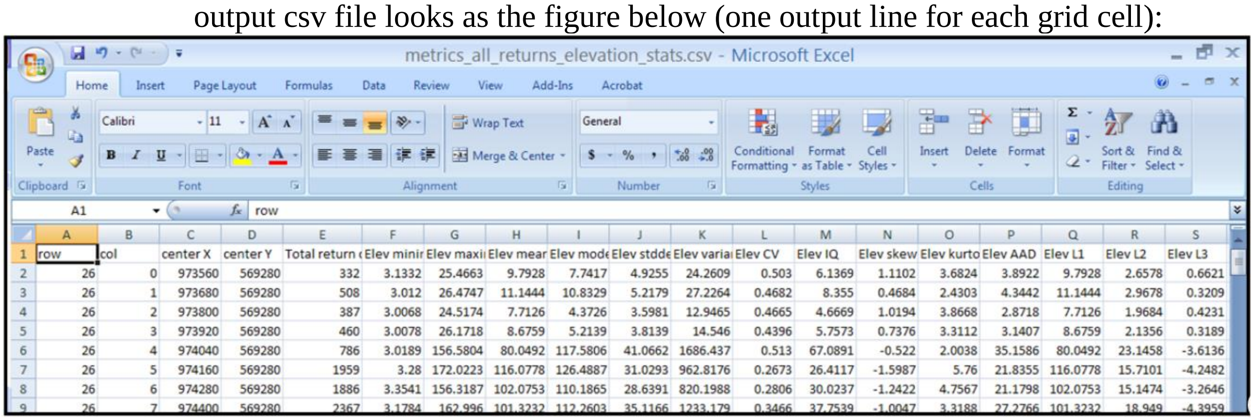
- % Canopy Cover ranges from 90% (plot 1) on the old growth site to 15% (plot 2) on the site that was treated with heavy thinning. Plot 3 falls somewhere in the middle, which makes sense. This is easily visualized when you inspect the overhead view of the point cloud. Remember the point clouds are colored by height, hence any blue is the ground below the canopy.
- The standard deviation is lowest in the lightly thinned stand (plot 3), this would indicate that the returns are more evenly distributed through the canopy. Could that be an indication of an evenly distributed vertical canopy structure? Take some time to explore some of the other metrics in outmetrics.csv. Can you find any interesting correlations? Refer to the CloudMetrics section of the Fusion manual for more explanation.

The **cloudmetrics** output is most often used with the output from the **ClipData** program to compute metrics (just as we did in part 1) that will be used for regression analysis. The next step in the regression analysis would be to explore relationships between field data recorded at the plots and the plot metrics we calculated using FUSION. Once these relationships are established using any number of analysis techniques (linear regression, random forest, etc….) you can apply the resulting equations across your whole study area. To do this you would need to compute the same metrics you did for the plots across the lidar acquisition. The **GridMetrics** FUSION command computes the same metrics as **CloudMetrics**, but the output is a **raster** (grid) format with each record corresponding to a single **grid** cell. In this part of the exercise, we will compute **GridMetrics** for the example data set.

10.6 Grid Metrics

1. Open DOS, navigate to c:\Fusion folder and type in gridmetrics to explore the full syntax.
2. Go to the GridMetrics section in the Fusion manual for more info.
3. Start a new Notepad document and save it as **grdmetrics.bat** in the Lab10 folder. Use the following switches and parameters to create your batch file:
    a. **C:\Fusion\Gridmetrics**: Fusion's command line executable
    b. **/minht:3** Switch to only use returns above a certain height to compute metrics. By choosing 3 feet we will exlude "ground" returns from our output stats.
    c. **/nointensity**: Switch for excluding intensity metrics.
    d. **T:\Teach\Temp\JaneDoe\FE444\Lab10\4800k_ground_surface.dtm**: File path for the bare-earth surface model used to normalize the LIDAR data (subtract the bare-earth surface elevation from each lidar point elevation).
    e. **12**: Parameter for a 12 foot height break for calculating cover metrics.
    f. **20**: Parameter for a 120 foot cell size, usually correlated with a fixed plot size.
    g. **T:\Teach\Temp\JaneDoe\FE444\Lab10\metrics.csv**: Output data file path. Note: we include the full file path. If the path is incorrect, or doesn't exist, the output file will not be created.
    h. **T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800k_data.las**: Input data file path. In this case we have used our single lidar tile. In a project scenario you will probably designate a number of tiles.
    i. Your syntax should look similar to the following:
    C:\Fusion\Gridmetrics /minht:3 /nointensity
    T:\Teach\Temp\JaneDoe\FE444\Lab10\4800k_ground_surface.dtm 12 120
    T:\Teach\Temp\JaneDoe\FE444\Lab10\metrics.csv
    T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800k_data.las

    j. In DOS navigate to **T:\Teach\Temp\JaneDoe\FE444\Lab10** and type in **grdmetrics** to run the command.
    k. After the program finishes, navigate to **T:\Teach\Temp\JaneDoe\FE444\Lab10**.
    l. Open the output file (**metrics_all_returns_elevation_stats.csv**) in Excel. The output csv file looks as the figure below (one output line for each grid cell):



	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S
1	row	col	center X	center Y	Total return (	Elev minir	Elev maxii	Elev mear	Elev mode	Elev stdde	Elev varia	Elev CV	Elev IQ	Elev skew	Elev kurto	Elev AAD	Elev L1	Elev L2	Elev L3
2	26	0	973560	569280	332	3.1332	25.4663	9.7928	7.7417	4.9255	24.2609	0.503	6.1369	1.1102	3.6824	3.8922	9.7928	2.6578	0.6621
3	26	1	973680	569280	508	3.012	26.4747	11.1444	10.8329	5.2179	27.2264	0.4682	8.355	0.4684	2.4303	4.3442	11.1444	2.9678	0.3209
4	26	2	973800	569280	387	3.0068	24.5174	7.7126	4.3726	3.5981	12.9465	0.4665	4.6669	1.0194	3.8668	2.8718	7.7126	1.9684	0.4231
5	26	3	973920	569280	460	3.0078	26.1718	8.6759	5.2139	3.8139	14.546	0.4396	5.7573	0.7376	3.3112	3.1407	8.6759	2.1356	0.3189
6	26	4	974040	569280	786	3.0189	156.5804	80.0492	117.5806	41.0662	1686.437	0.513	67.0891	-0.522	2.0038	35.1586	80.0492	23.1458	-3.6136
7	26	5	974160	569280	1959	3.28	172.0223	116.0778	126.4887	31.0293	962.8176	0.2673	26.4117	-1.5987	5.76	21.8355	116.0778	15.7101	-4.2482
8	26	6	974280	569280	1886	3.3541	156.3187	102.0753	110.1865	28.6391	820.1988	0.2806	30.0237	-1.2422	4.7567	21.1798	102.0753	15.1474	-3.2646
9	26	7	974400	569280	2367	3.1784	162.996	101.3232	112.2603	35.1166	1233.179	0.3466	37.7539	-1.0047	3.3188	27.2766	101.3232	18.949	-4.3959

4. Let's note a few things about the output CSV file.
    a. You should notice that each row has a column denoting the center x and center y for that grid cell.
    b. As you scroll through the file you may see some rows that have –9999 values, this is FUSION's nodata value.

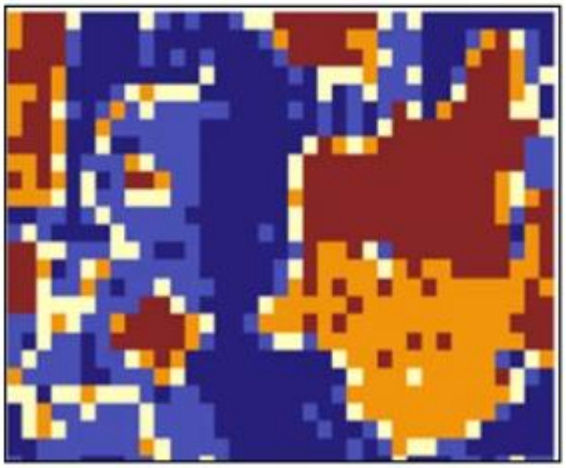
c. Cloudmetrics and GridMetrics produce the same metrics. This capability in FUSION makes it possible to model relationships observed between lidar plot metrics and field plot measurements and then apply them across the entire lidar acquisition. Any of the rows in the CSV file can be converted into ASCII files and exported into a GIS for analysis/modeling. Let's export one of the metrics (columns) now.

10.7 OPTIONAL. Prepare the CSV2GRID command to export one of the columns and create an asci file representing the % Canopy Cover Metric

1. Open DOS and navigate to **c:\Fusion.** Then type **CSV2Grid** to study the syntax.
2. Go to the **CSV2Grid** section of the Fusion Manual for more info.
3. Let's create our script to extract the metrics from our study area. Start a new Notepad (.txt) document and save it as **covergrd.bat** in the lab 10 folder. Use the following switches and parameters to create your batch file:
    a. **C:Fusion\csv2grid** Fusion's command line executable
    b. **T:\Teach\Temp\JaneDoe\FE444\Lab10\metrics_all_returns_elevation_stats.csv** InputDataFile, our output from the Gridmetrics command
    c. **49** denotes column 49 in the gridmetrics csv output file. Column 49 correlates to the % cover calculation for each cell or "Percentage first returns above 12.00".
    d. **T:\Teach\Temp\JaneDoe\FE444\Lab10\cover_grid.asc** Output Data File, output grid in ascii format. This will allow us to use the data in a GIS.
    e. Your syntax should look similar to the following:

C:Fusion\csv2grid T:\Teach\Temp\JaneDoe\FE444\Lab10\metrics_all_returns_elevation_stats.csv 49 T:\Teach\Temp\JaneDoe\FE444\Lab10\cover_grid.asc

    f. In DOS navigate to **c:\fusion** and copy the command above then execute it or navigate to T:\Teach\Temp\JaneDoe\FE444\Lab10 and double click the **covergrd.bat** file.
    g. Now open **ArcMap**, navigate to and load your new ASCII file (**cover_grid.asc**). Ignore the spatial reference warning. *Note*: In the future you will want to define the projection of your lidar derivatives, which will be the same as the raw lidar data. ArcMap does not recognize FUSION's projection so you have to manually redefine it!
    h. In the Search bar type in **Calculate Statistics** and run that tool for **cover_grid.asc** (there is no need to define an area of interest).
    i. Right click on **cover_grid.asc** in the index and select **Properties** from the drop down menu.
    j. Under the **Symbology** tab, select **Classified** and your desired color ramp. Click **Apply**. Your ArcMap display should now look somewhat similar to the following graphic.



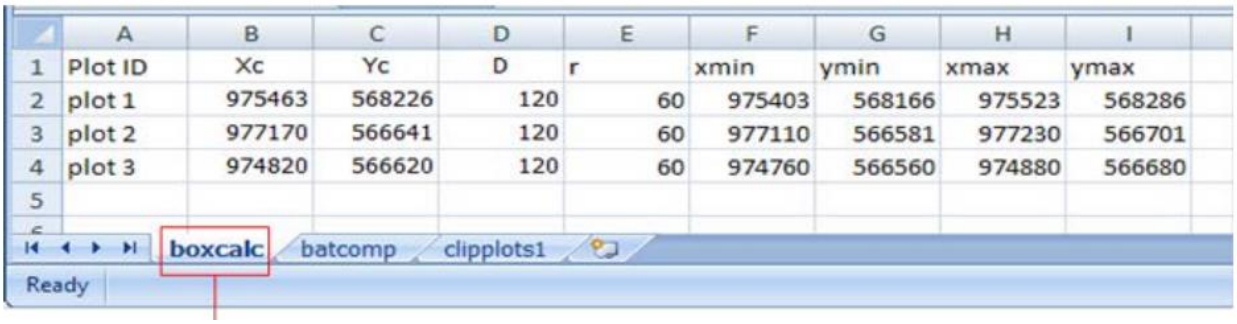
k. Add the lidar reference image, **orthophoto_4800K.jpg**, to your map display.
l. Explore both layers together for a few moments. Based on your image interpretation skills does the **% Canopy Cover** lidar derivative layer correlate with the photo. Do you think this would be a good GIS layer for your whole forest?!
m. Any of the metrics calculated in the **Gridmetrics** CSV output file can be exported into grid format and added to your GIS for analysis.

10.8 Appendix: Create an Excel spreadsheet

A very effective method for creating a batch file is by using MS Excel. The advantage of creating such a file in a spreadsheet is the facility to generate series of incremented field entries and to be able to concatenate several fields into a single text string. We're only extracting LIDAR subsets for three plots in order to keep this exercise relatively quick and manageable. However, the real value of this batching process will only become obvious when you want to extract a large number of LIDAR subsets. The three plot centers (made up for the sake of this exercise) have the following locations (Xc, Yc in the same units and coordinate system as the lidar data) and they have a diameter (D) of 120 meters shown below in the table:

Xc	Yc	D
975463	568226	120
977170	566641	120
974820	566620	120

1. Open Excel
2. Name the first Worksheet tab **boxcalc**
3. Name the second Worksheet tab **batcomp**



	A	B	C	D	E	F	G	H	I
1	Plot ID	Xc	Yc	D	r	xmin	ymin	xmax	ymax
2	plot 1	975463	568226	120	60	975403	568166	975523	568286
3	plot 2	977170	566641	120	60	977110	566581	977230	566701
4	plot 3	974820	566620	120	60	974760	566560	974880	566680
5									

4. Create column headings labeled: **Command, Input File, and Output File**.
5. Command Column: this contains the path to and name of the fusion command line executable— in this case **clipdata**.
6. **Input File** Column: contains the path and name of the lidar data set. This example contains only one file from which data will be subset (**lda_4800K_data.las**) but more can be used as required.
7. **Output File** Column: contains the path and filename of the output file (the plot subsets). They were built by concatenating the path, the plot-id listed in the boxcalc worksheet and the extension of the file: =CONCATENATE("b.
   T:\Teach\Temp\JaneDoe\FE444\Lab10",boxcalc!A2,".las")
8. The values in the next four columns are the column headings and min and max values for the bounding box copied from the boxcalc worksheet.
9. Referencing a cell from another worksheet in the same spreadsheet is done by adding the name of the worksheet followed by an exclamation mark: ex: =boxcalc!F2
10. The final column will be labeled Full Syntax which will be a concatenation of all of the previous columns and the insertion of the /shape:1 switch=CONCATENATE(A2," /shape:1 ",B2," ",C2," ",D2," ",E2," ",F2," ",G2)

The results for the first row of the Full Syntax Column should look like:
..\clipdata /shape:1 b. T:\Teach\Temp\JaneDoe\FE444\Lab10\lda_4800K_data.las c b.
T:\Teach\Temp\JaneDoe\FE444\Lab10\clipplot1.las 975403 568166 975523
568286

11. Next, copy Column H (the Full Syntax Column) and paste its values (Paste Special | Values) in another worksheet named bat:
12. Save your Excel spreadsheet (any name and location that makes sense to you) but don't close the spreadsheet.
13. Save the bat worksheet, which should look like the following graphic (it should be a text file named clipplots1.bat in your batch file directory).

14. Type clipplots1 in the command prompt and hit enter to let it run.
15. Using Windows Explorer, navigate to the output folder directory (T:\Teach\Temp\JaneDoe\FE444\Lab10) and verify that the files clipplot1.las, clipplot2.las and clipplot3.las were created.

10.9 Questions

1. How many points are stored in each of the three plot las files?
2. What is the area of plot #2?
3. What is a normalized points cloud?
4. Do you need to normalize the point cloud if you plan to measure the height of the trees using Fusion? Justify your answer with 2-3 sentences at the most
5. What is the average height of plot #1?
6. What is the height of the tallest tree in plot #1, 2, and 3?
7. What is the height of the smallest tree in plot #1?
8. What are the factors affecting tree heights from lidar point cloud? (look at the notes)
9. Is the height computed from lidar overestimating or underestimating the tree height?
10. What is the expected accuracy of height estimated from lidar point cloud?