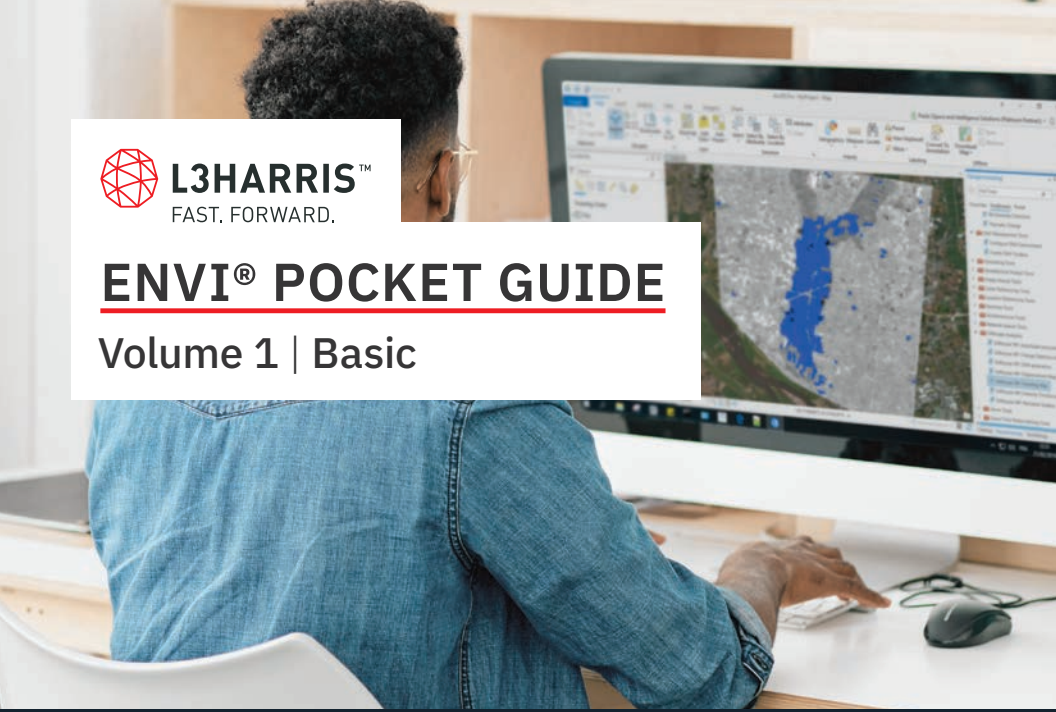


L3HARRIS™
FAST. FORWARD.

ENVI® POCKET GUIDE

Volume 1 | Basic

L3HarrisGeospatial.com/ENVI

Authored as a Derivative from ENVI Bootcamp & ENVI Documentation Center By:
CW3 Augustus Wright, 125D EN

In association with the U.S. Army TWI Program.

CONTENTS

The ENVI® Pocket Guide is a quick reference booklet not intended to be read from cover to cover although it can be. The intent is to provide users with succinct steps on how to accomplish common tasks in ENVI.

If you need or desire comprehensive explanations of tasks from this guide refer to the following resources:

ENVI Documentation Center
l3harrisgeospatial.com/docs

ENVI Tutorials
l3harrisgeospatial.com/docs/tutorials.html

ENVI Videos
l3harrisgeospatial/Learn/Videos

ENVI Help Articles
l3harrisgeospatial.com/Support

Tech Support
(+1)303-413-3920

Email Support
geospatialinfo@L3Harris.com

ENVI® INTERFACE KEY

Some numbers represent GUI groups separated by commas from left to right.

1. Open file, Data Manager, Chip to PowerPoint (Report Generation)

2. Scroll View, Cursor Value, Select, Pan, Fly, Rotate

3. Zoom, Fixed Zoom In/Out, Full Extent, Percent/Ratio

4. North Up, Type or Preset Angle

5. Arbitrary/Spectral Profile, Scatter Plot, ROI, Feature Count

6. Digitizing, Vectors, Enhanced Annotations

7. Go To Coordinate

8. Brightness

9. Contrast

10. Histogram Stretch

11. Sharpen

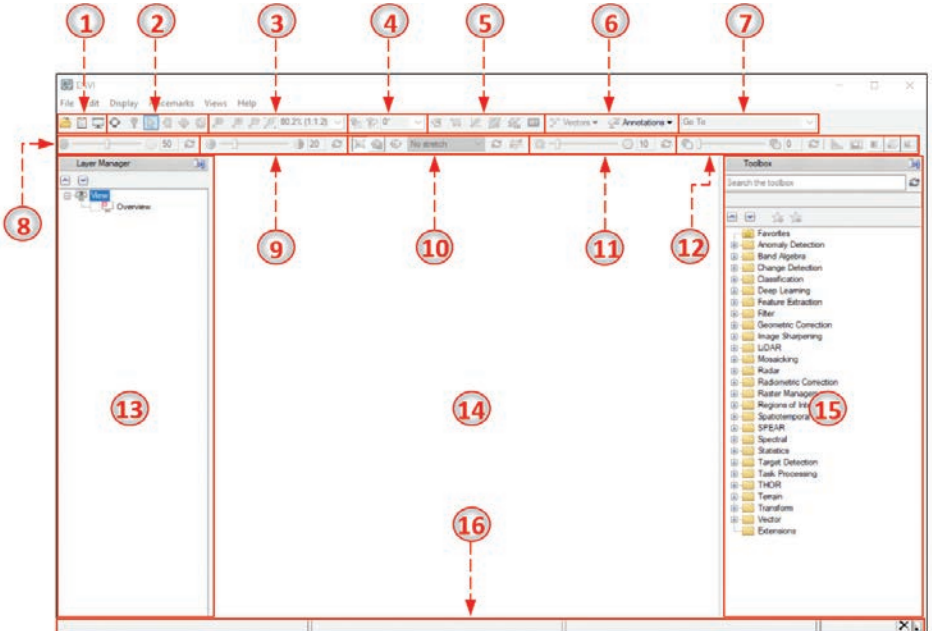
12. Transparency, Mensuration Portal, Blend, Flicker, Swipe

13. Layer Manager

14. Data Viewer/Image Window

15. Toolbox

16. Cursor Value, Status Bar


1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
ENVI
File Edit Display Bookmarks Views Help
Layer Manager
View
Overview
Toolbox
Search the toolbox
Favorites
Anomaly Detection
Band Algebra
Change Detection
Classification
Deep Learning
Feature Extraction
Filter
Geometric Correction
Image Sharpening
LiDAR
Mosaicking
Radar
Radiometric Correction
Raster Management
Regions of Interest
Spatiotemporal
SPEAR
Spectral
Statistics
Target Detection
Task Processing
THOR
Terrain
Transform
Vector
Extensions

GETTING STARTED

OPENING ENVI

1. On Windows, click Start > All Programs > ENVI 5.x > ENVI for ArcGIS® > ENVI 5.x (32-Bit). This method allows you to interface with ArcMap.

2. To open ENVI in 64-bit mode, click Start > All Programs > ENVI 5.x > 64-Bit > ENVI 5.x (64-Bit). This method does not allow you to interface with ArcMap.

3. To open ENVI programmatically via IDL, click Start > All Programs > IDL 8.x > IDL 8.x. The IDL window appears. In the IDL Console, type the following code: `IDL> E = ENVI()`. Then press the Enter key and ENVI will soon launch.

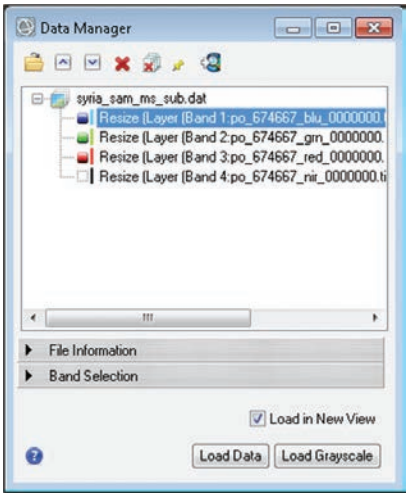
This is an introduction to familiarize users with common methods for opening ENVI, loading data, navigating, and performing stretches.

LOADING & REMOVING DATA

4. To load data, click the Open File icon . An explorer window will open. Navigate to the disk location of the file you wish to open.

5. Select the file and click Open. ENVI's native file format is .dat, but ENVI recognizes and reads most common formats such as: *.img, .ntf, rpf, .tiff, .las, .dt, dppdb, .sid, .shp, .evf, .jp2*, and many more.

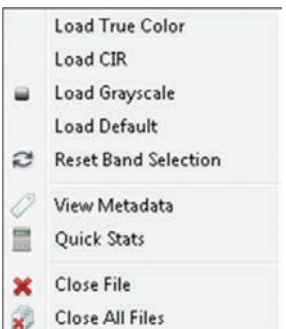
6. Another method is to click the Data Manager icon . The Data Manager appears, as seen on the previous page:



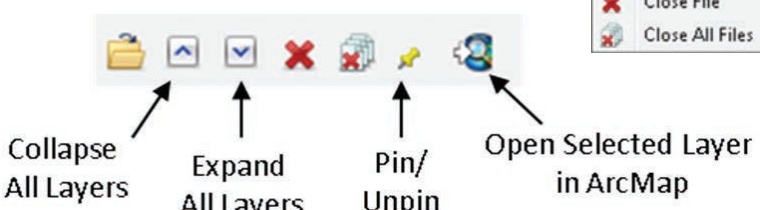
Windows Data Drag & Drop Functionality is fully supported in ENVI.

7. Load additional layers by clicking in the Data Manager. Explore different band combinations within the current view or a new view by clicking Load Data.

8. If wavelength information is available for the bands of an image, right-click on an image layer in the Layer Manager to load true-color or color-infrared (CIR) band combinations; for example:

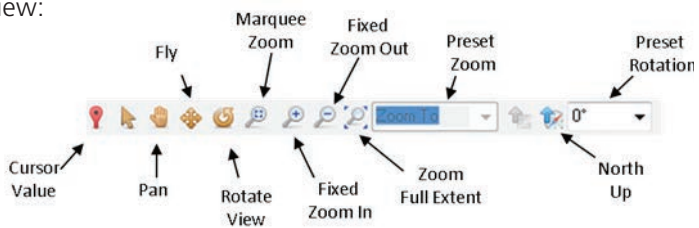


9. The Data Manager provides other data-management options:

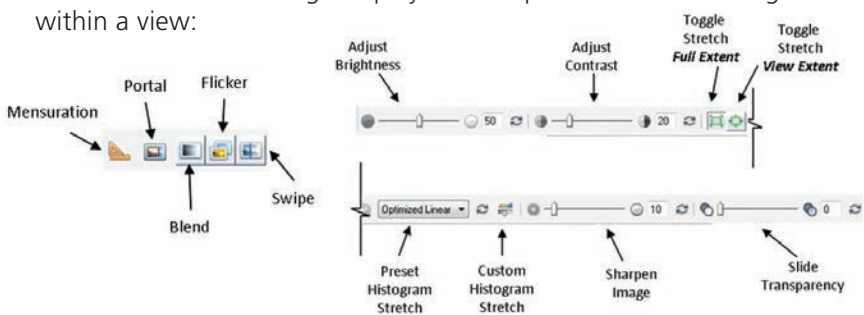


NAVIGATING WITHIN A VIEW

10. Use ENVI's intuitive navigation tools to perform the following within a view:



11. Use ENVI's intuitive image display tools to perform the following within a view:

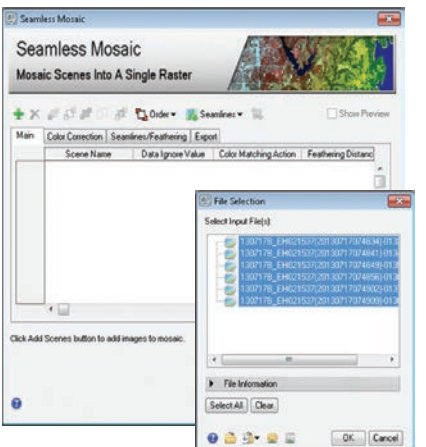


MOSAIC

SEAMLESS MOSAIC

1. From the Toolbox, select Mosaicking > Seamless Mosaic. The Seamless Mosaic panel appears. You will set all mosaic options within this single panel. Click the Help button in the lower-left corner of the panel if you have further questions.

2. Click the Add Scenes button  .



Before beginning, some imagery requires preparation such as cropping to remove distorted edges to ensure optimal output results.

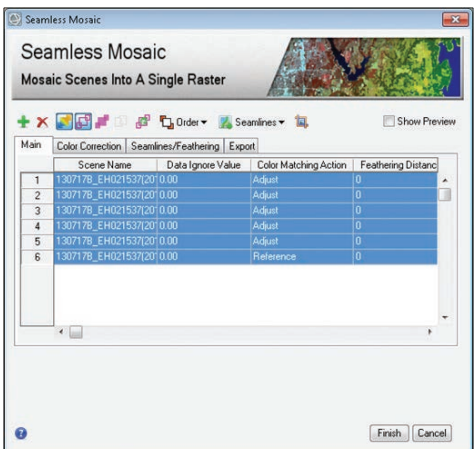
3. Click the Open File button in the Data Selection dialog. A file selection dialog appears.

4. Navigate to the folder containing the image data, and use the Ctrl or Shift key to select the files you need to mosaic. Click Open. The Data Selection dialog lists all selected scenes.

5. Click the Select All button.

6. Click OK.

7. The Seamless Mosaic dialog should look similar to the example on this page. Highlight all images using the main column.



8. Right-click the Data Ignore Value column header and select Change Selected Parameters to set your value. In most cases None will be used.

9. Highlight all images, then right-click and select Feathering Distance. Select Change Selected Parameters to set the value between 10 and 20 pixels or more, depending on image quality and resolution.

10. Click the Color Correction tab and toggle Histogram Matching. Accept the default of Overlap Area Only if you want to compute statistics from areas where the reference image overlaps with adjusted images. When there is at least 10% overlap between scenes, histogram matching based on statistics from overlapping areas performs better than statistics from the entire scene. Select Entire Scene if you want to compute the entire image. Go back to the Main tab and set your desired reference image under the Color Matching Action column. All others will be matched.

11. Click the Seamless drop-down arrow and select Auto Generate Seamlines.

12. Toggle the Show Preview check box. The system may take a few minutes to generate a preview, depending on your system specifications.

13. Use the Zoom tools to visually inspect the image before exporting it.

14. If you are not satisfied with the previewed result, go back to the Main tab and adjust the reference image under the Color Matching Action column. Adjust the Feathering Distance values if needed. If the "None" Data Ignore Value doesn't completely eliminate edge noise, you may need to crop the input image(s) in question before mosaicking.

15. If you are satisfied with the preview, proceed to the Export Tab. Optionally, use the Define Output Area tool before exporting to crop and designate a perfectly square or rectangle output.

16. Select ENVI or TIFF output format.

17. Browse to an Output Directory and name the output file.

18. Enable the Display result check box if you wish to display the result after exporting.

19. Enter the Date Ignore Value that you defined in the Main tab. If you want more control over the pixels of the output, you may enter a different value. However, the value must be within the range of the input scenes. Click the Help button for more details.

20. Choose the desired Resample Method.

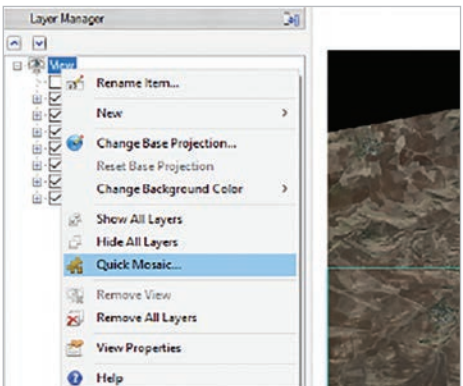
21. Click the Select Output Bands button if you wish to specify the bands to export. By default, all bands are selected.

22. Click Finish.

QUICK MOSAIC

METHOD 1: LAYER MANAGER

1. If your rasters are loaded in ENVI, right-click on the View layer in the Layer Manager, and select Quick Mosaic. The Quick Mosaic dialog opens.



As its name suggests, Quick Mosaic quickly creates a virtual raster from multiple overlapping or adjacent georeferenced rasters. It allows users to evaluate multiple rasters using a single raster without going through the steps and processes of a Seamless Mosaic.

2. By default, all of the rasters in the Layer Manger are selected in the Input Rasters field. Use the Ctrl or Shift key to deselect any individual rasters that you do not want to include in the mosaicking process.

3. Enter the values of the border pixels in the Data Ignore Value field. Those pixels will be ignored and become transparent in the virtual mosaic. You may also leave the Data Ignore Value field blank.

4. Enter a name in the Mosaic Name field or accept the default.

5. Accept the Yes default for the Remove Layers option if you want to remove the input rasters from the Layer Manager after the virtual raster is created. Select No if you want to keep the input rasters.

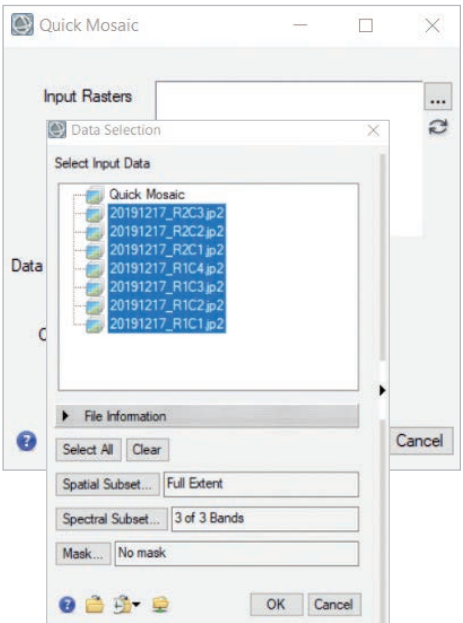
6. Click OK.

METHOD 2: ENVI TOOLBOX

1. Select Mosaicking > Quick Mosaic.

2. Click the Browse button ... in the Input Rasters field. The Data Selection dialog opens.

3. If the rasters to be mosaicked are loaded in ENVI, they will appear in the Select Input Data field in the Data Selection dialog. Click the Select All button or use the Ctrl or Shift key to select your input rasters.



4. If the rasters to be mosaicked are not listed in the Select Input Data field, click the Open File button to navigate to the rasters' directory.

5. Use the Ctrl or Shift key to select the rasters to be mosaicked. Click Open, then click OK to close the Data Selection dialog.

6. The Quick Mosaic dialog should look similar to the example on the next page.

7. Enter the value of the background or border pixels in the Data Ignore Value field, or leave it blank.

8. For the Output Raster option, accept the Virtual Raster default if you want a memory-only output. The virtual raster will be created and displayed, but it will not be saved. Choose File if you want to save the output.

9. When you choose File, you will be prompted to navigate to an output directory. Name your output raster, then click Save.

10. Back in the Quick Mosaic dialog, ensure the Display result check box is enabled.

11. Click OK.

SUBSET

1. Click the ROI Tool icon and draw a square or polygon over the extent of the image you want to subset. Double-click to finish drawing the shape. Name the ROI you just created or leave it set to the default name of ROI #1.

2. From the Toolbox, select Regions of Interest > Subset Data from ROIs.

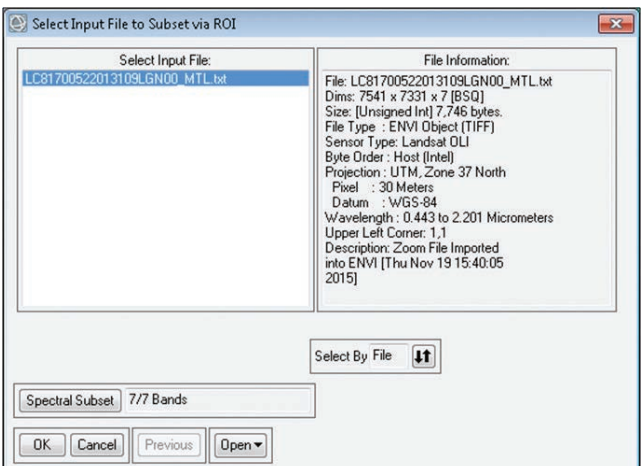
3. Under Select Input File, highlight the image that will be subset.

Subsets will be performed using Regions of Interest (ROI). Supported ROI inputs include .xml files created with the ROI Tool, .shp files, and ENVI Classic .roi and .evf files.

4. Under Select Input File, highlight the image that will be subset.

5. The Select Input File to Subset via ROI dialog also gives you the option to Select By: File or Band using this button: ⇵

   You can also specify bands by clicking the Spectral Subset button.



6. Accept all defaults and click OK.

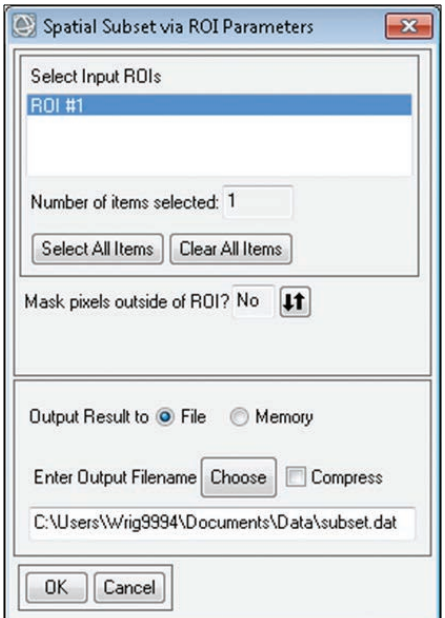
7. Highlight ROI #1 on the Spatial Subset via ROI Parameters dialog.

8. Enter an Output Filename by clicking Choose.

9. Navigate to desired output directory and name the file something like "subset.dat" as seen in the example.

10. Click OK.

11. The newly created subset will be added to your View once processing is complete.

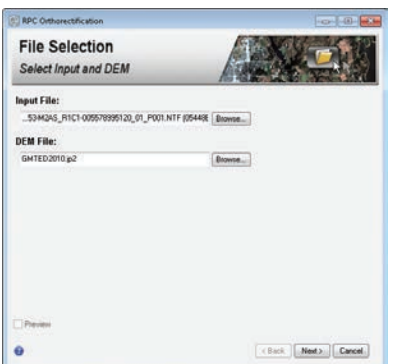


ORTHORECTIFY

1. Open the image you will orthorectify, along with the associated DEM if you have one.

2. From the Toolbox, select Geometric Correction > Orthorectification > RPC Orthorectification Workflow. The File Selection dialog appears.

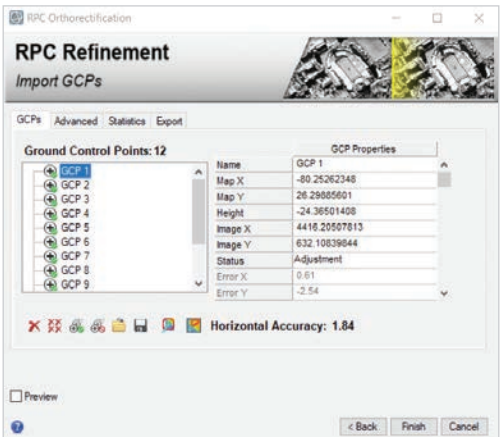
3. Click Browse to select an image and DEM. If you do not have a DEM, use the default GMTED2010.jp2 file provided by ENVI.



To perform orthorectification, the input image must have Rational Polynomial Coefficients (RPCs). Review the metadata to ensure this information is present. If you do not have elevation data, ENVI provides a default 30 arc-second (1 km) DEM called GMTED2010.jp2 as part of the software installation.

4. Click Next to proceed to RPC Refinement.

5. Click the GCPs tab if you have Ground Control Points (GCPs) to add. GCPs improve accuracy. Click the Load GCPs button to load the GCPs. The GCPs appear in the Ground Control Points field. Additionally, the Horizontal Accuracy value is adjusted from a value of 0 (zero), indicating the orthorectification result will have better accuracy.

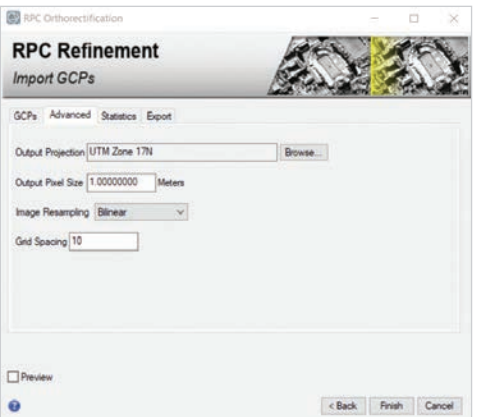


6. Take a moment to inspect the GCPs. Click on the GCPs in the Ground Control Points field and evaluate their properties in the GCP Properties field. Click the Help button for more details.

7. Click the Advanced Tab.

8. The Output Projection defaults to the coordinate system for the orthorectified image. To change it, click Browse and select a different coordinate system.

9. The Output Pixel Size defaults to the value derived from the pixel size of the input image. To change it, enter an Output Pixel Size in the same unit as the selected output projection.



10. Select an Image Resampling technique.

11. Enter a Grid Spacing value. The default value is 10.

12. Enable the Preview check box if you want to preview the orthorectified result before processing.

13. Click the Statistics tab. Accept the default settings. Click the Help button for details on GCP Statistics, Horizontal and Vertical Accuracies, and Error Overlay Threshold.

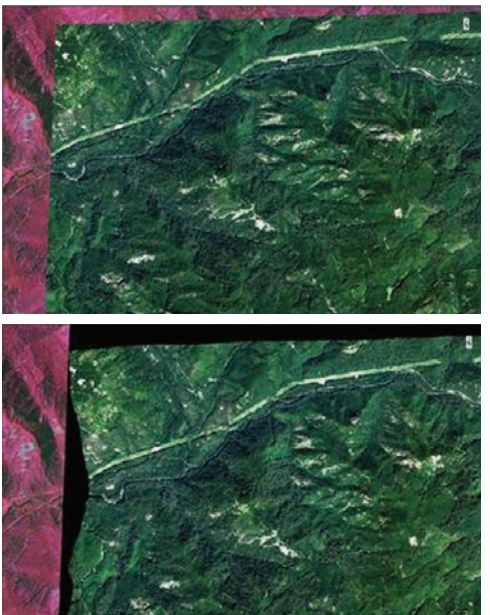
14. Click the Export Tab.

15. Choose an Output File format.

16. Click Browse and navigate to an output directory. Name the output file, then click Save.

17. Enable the Preview check box to open a Portal View. Use the Portal View to visually inspect the results.

18. Click Finish.

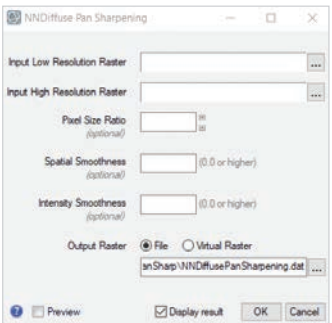


Before and after ortho-examples.

PANCHROMATIC SHARPEN

METHOD 1: NNDIFFUSE PAN SHARPENING

1. Open your low resolution multi-spectral and high-resolution panchromatic images in an ENVI view.

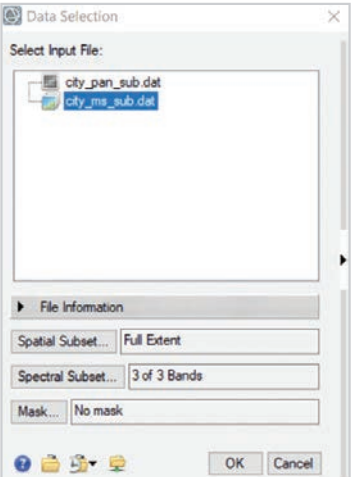


Panchromatic, or "pan" sharpening is an image merging process that allows users to create a new image by combining high-resolution panchromatic images with low-resolution, multispectral images to produce a color image for literal analysis. ENVI provides multiple methods. Only two have been selected for this guide to provide a basic understanding of the process. Proceed with Method 1 if your images do not require co-registration or atmospheric correction. Otherwise use Method 2.

2. In the ENVI Toolbox, search for "NNDiffuse," or select Image Sharpening > NNDiffuse Pan Sharpening. The NNDiffuse Pan Sharpening dialog appears.

3. Click the Browse button ... next to the Input Low/High Resolution Raster fields, then select multispectral and panchromatic images, respectively.

4. Use the marque zoom tool to zoom into any area.

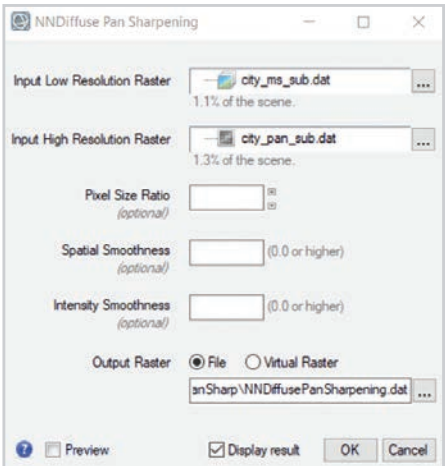


5. Enable the Preview check box in the NNDiffuse Pan Sharpening window. A new view displays a preview of the pan-sharpened result. This allows you to visually inspect the before-and-after results before processing the data.

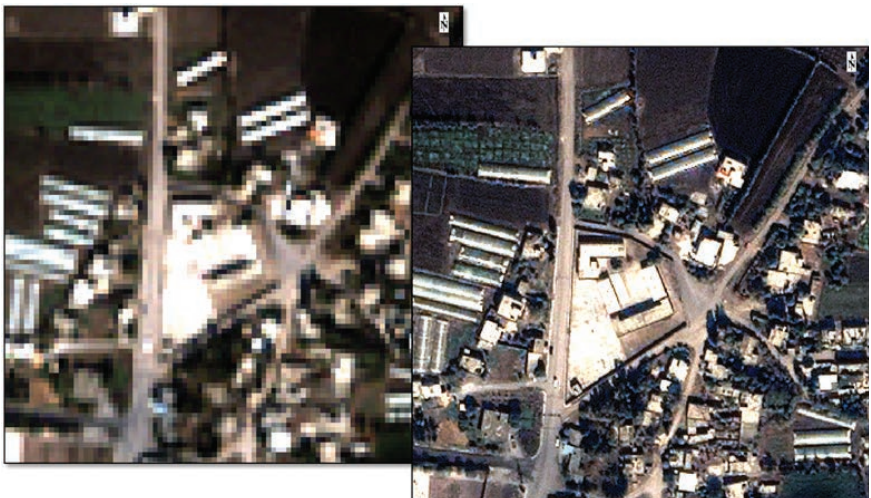
6. If you are satisfied with the Preview results, select an Output Raster option. Choose File if you want to save the output. Click the Browse button ... and navigate to an output directory. Name the output file, then click Save. Choose Virtual Raster if you want a memory-only output.

7. Enable the Display result check box if you want to display the result after processing.

8. Click OK.



Before and after NNDiffuse pan sharpen examples:

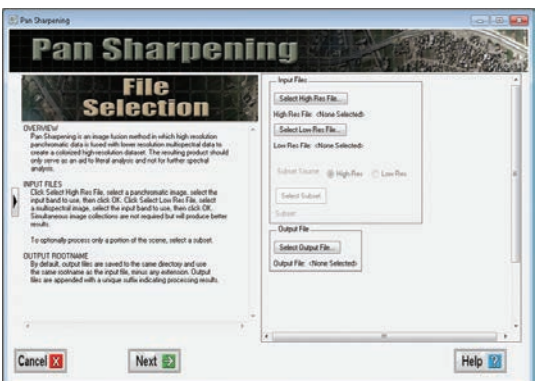


METHOD 2 : SPEAR PAN SHARPENING

1. Open the high-resolution panchromatic and low-resolution multispectral images in ENVI.

2. In the ENVI Toolbox, select SPEAR > SPEAR Pan Sharpening. The Pan Sharpening workflow appears.

3. Click the Select High Res File button. Select a panchromatic image and click OK.

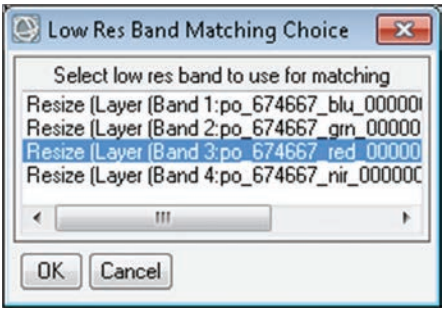


In this workflow, images will be co-registered and atmospherically corrected before proceeding with pan sharpening.

4. Click the Select Low Res File button. Select a multispectral image and click OK.

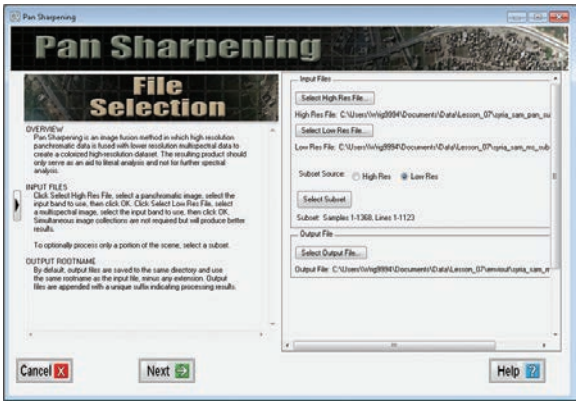
5. The Low Res Band Matching Choice dialog appears. A best practice is to choose Band 3 because spectral properties in the red band closely match those of panchromatic images, which facilitates co-registration. Click OK.

6. Click Select Output File, navigate to an output directory and name the file *_pansharp.dat.



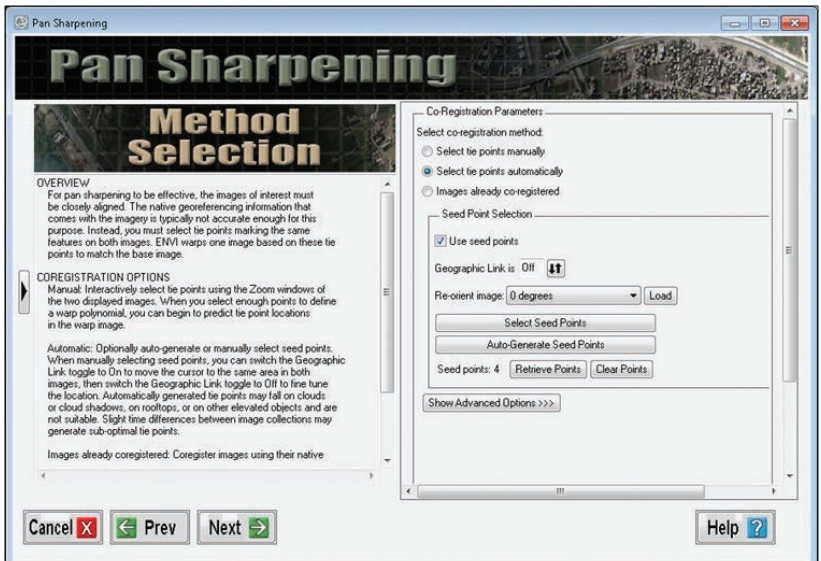
7. The File Selection panel of the workflow should resemble the this example.

8. Click Next. In the following window, enable Select tie points automatically.



9. Ensure that Use seed points is enabled, and click Auto-Generate Seed Points. This allows ENVI to estimate GCPs. Alternatively, you can choose your own by clicking Select Seed Points.

10. The Method Selection panel should resemble the following example:



11. Click Next. A series of windows will open in addition to the Pan Sharpening workflow. Bring the Image to Image GCP List forward.

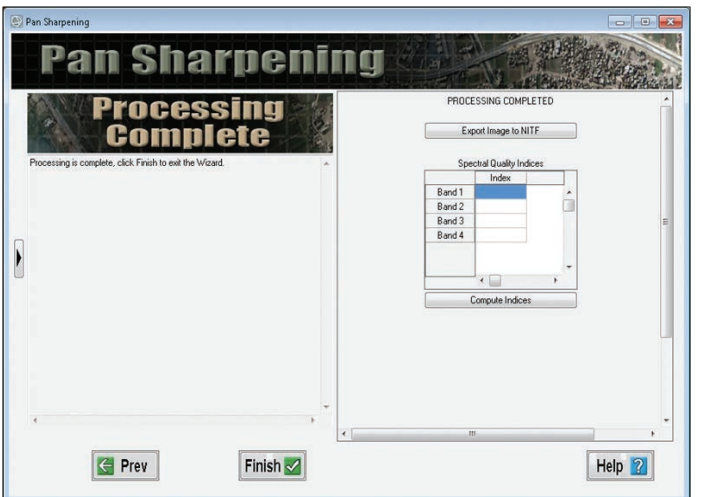
12. Click Options > Order Points by Error. The seed points with the highest RMS Error are brought to the top of the list.

13. Highlight seed points with an RMS Error higher than 3, and click Delete.

14. Bring forward the Pan Sharpening workflow, and set the Maximum allowable RMS per GCP value to 1.0. Click Apply.

15. Choose the desired Warp Parameters. For this exercise, set Method to "Polynomial" and Interpolation to "Cubic Convolution." Click Next.

10. In the final panel of the workflow, select a Sharpening Method. For this exercise, select Gram-Schmidt and click Next. Once processing is complete, click Export the image to NITF. Or, select File > Save As in the main window of the three-window display group. Then save the result to an image file or postscript file.



CHANGE DETECTION

1. Click the Open File button  in the ENVI toolbar, and open two images over the same geographic region on different dates.

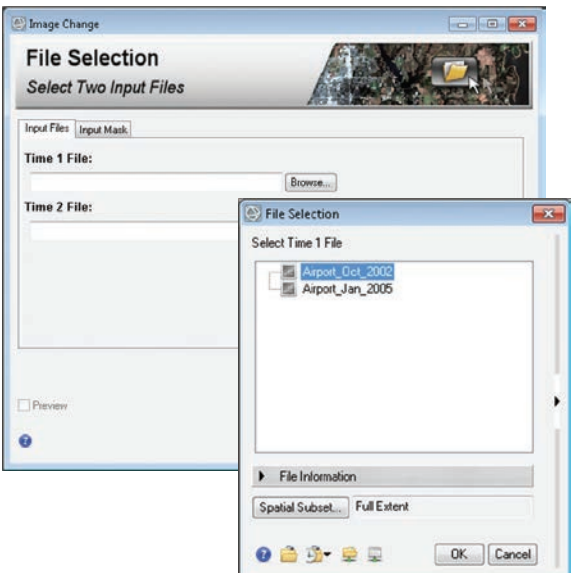
2. In the ENVI Toolbox, select Change Detection > Image Change Workflow. The Image Change workflow appears.

3. Click Browse next to Time #1 File and select the oldest/first image. Click OK.

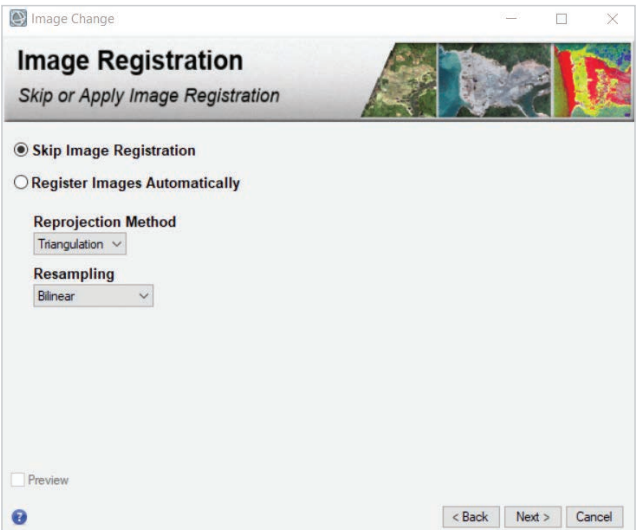
The steps in this section will produce a 2-Color Multi View (2CMV) Change Detection result. The end product will be a standard red and blue image based on changes that have taken place between images captured on two separate dates. Remember the phrase "red has fled" and "blue is new." This exercise assumes the images are already co-registered. If your images are not co-registered, you have the option to automatically co-register them during the workflow.

4. Click Browse next to Time #2 File and select the newer/second image. Click Next.

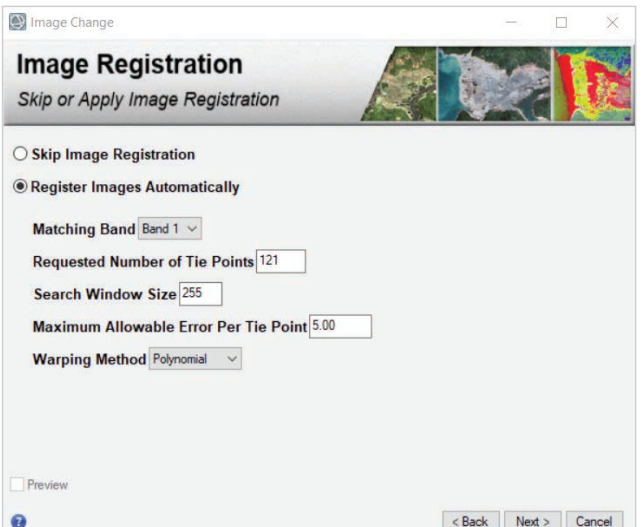
5. The Image Registration panel appears. If your images require co-registration, toggle Register Images Automatically. Otherwise, toggle Skip Image Registration.



6. For Skip Image Registration, you can accept the defaults for the Reprojection Method and Resampling technique or select different options from the drop-down lists.



7. For Register Images Automatically, ENVI applies the registration values and settings based on the input files. You can accept the default values and settings or change them to have more control over the process. Click the Help button for details.



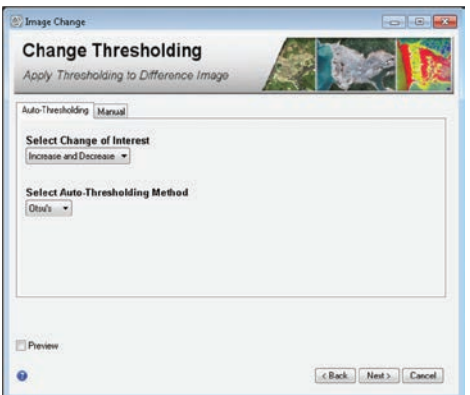
8. Click Next. The Change Method Choice panel appears.

9. Accept the default Image Difference and click Next.

10. Accept the default Difference of Input Band and Band 1 under Select Input Band. Click Next. The Thresholding or Export panel appears.

11. Accept the default Apply Thresholding and click Next. The Change Thresholding panel appears. This panel lets you enter a manual threshold or use an Auto-Thresholding technique.

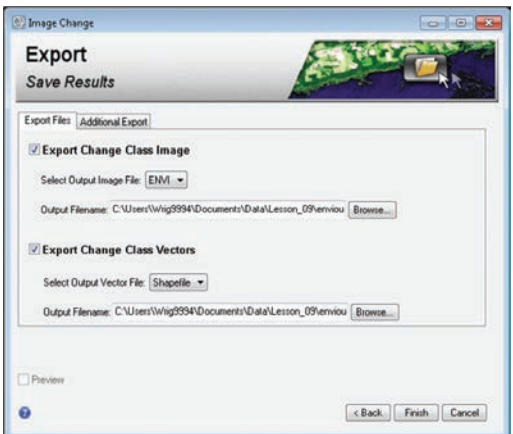
12. Under Select Change of Interest, you can choose Increase and Decrease, Increase Only, or Decrease Only. Increase and Decrease will return both red and blue 2CMV results. Red indicates decrease and blue indicates increase ("red has fled, blue is new").



13. Under Select Auto-Thresholding Method, accept the default selection of Otsu's. For a comprehensive explanation of Otsu's, Tsai's, Kapur's, and Kittler's thresholding methods, refer to the ENVI documentation. Click Next. The Cleanup panel appears.

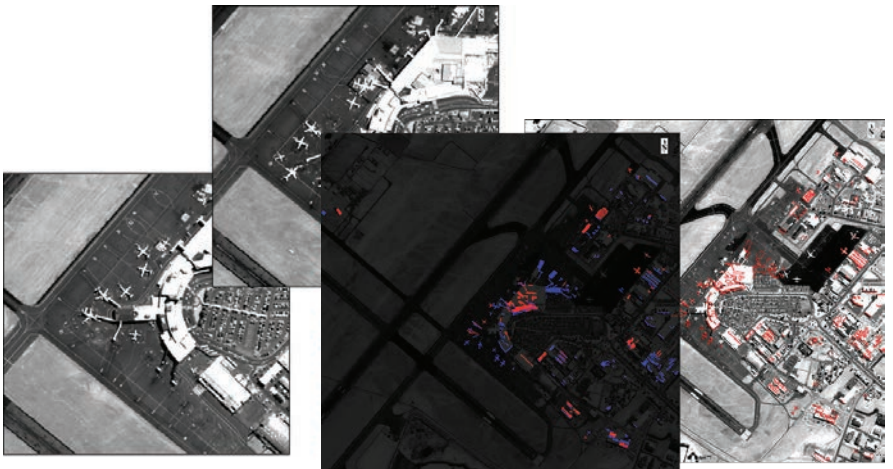
14. The Cleanup panel lets you refine the results. Accept the defaults and click Next, or toggle Preview and manually adjust the previewed result to your liking. After clicking Next, the Export panel appears.

15. The Export panel lets you save both image and vector outputs. Click Browse on both the Export Change Class Image and Export Change Class Vectors. Navigate to your output directory and name the output files appropriately. Click Open, then click Finish.



16. Use the transparency slider to visually inspect the results.

Old image, new image, change detection examples:



ANOMALY DETECTION

1. Open an image in ENVI through the Data Manager or by dragging and dropping from Windows Explorer or ArcCatalog.

2. From the Toolbox, click Anomaly Detection > Anomaly Detection Workflow. The Anomaly Detection wizard will appear. Click Next.

Anomaly detection is a process performed on imagery to highlight significant or unnatural spectral differences in pixels versus neighboring values. Anomaly detection is useful in determining things such as disturbed earth or spikes in energy levels versus surroundings, for example. This process is useful in narrowing down searches to determine if and where further investigation is warranted.

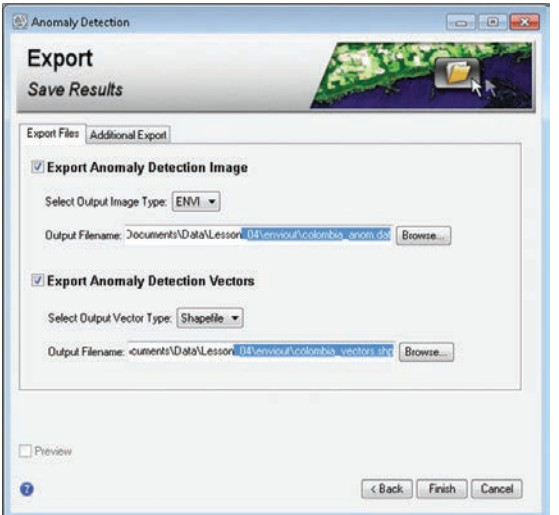
3. Select the RXD-UTD algorithm under Anomaly Detection Method. For a detailed explanation of this and other algorithms for this workflow, refer to the ENVI documentation.

4. Accept Global under Mean Calculation Method to process the spectrum of the full dataset. Select Suppress Vegetation if your image contains a lot of vegetation. Click Next.

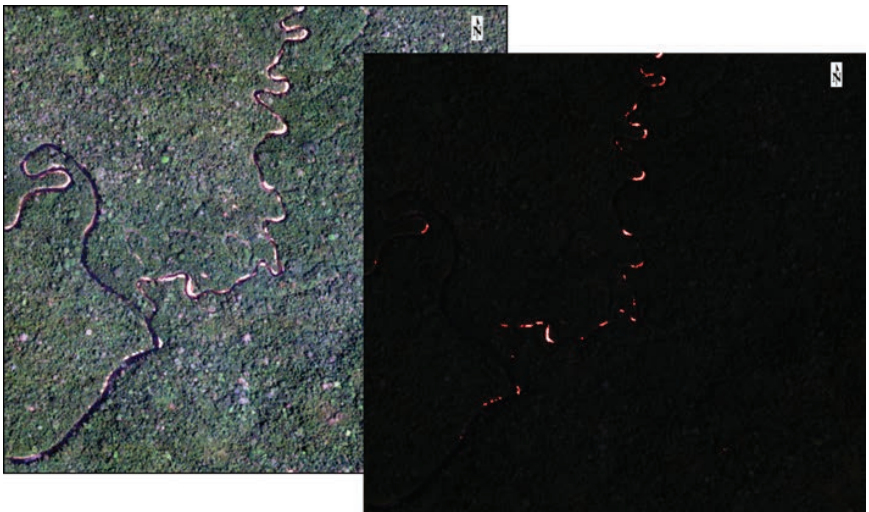
5. The default value for Anomaly Percentage Threshold is 0.05. Accept this value, or adjust it using trial-and-error. The higher the threshold, the more data will be perceived as defects. Experiment with the Preview check box. Click Next. The Export panel appears.

6. In the Export panel, ENVI will automatically append the image and vector output files to the root name. ENVI will also save the output files in the same directory as the input file. You can accept these defaults or change them by clicking Browse. Click Finish.

7. The results will automatically open in ENVI for further investigation and/or exploitation. White pixels in an image indicate anomalies. Red in a vector file indicates an anomaly.



Input image, anomaly image overlayed with anomaly vector examples:

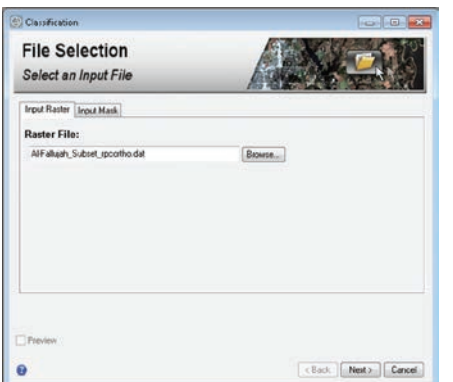


TERRAIN CATEGORIZATION

1. Open a multispectral image in ENVI for classification.

2. In the ENVI Toolbox, click Classification > Classification Workflow. The File Selection panel appears.

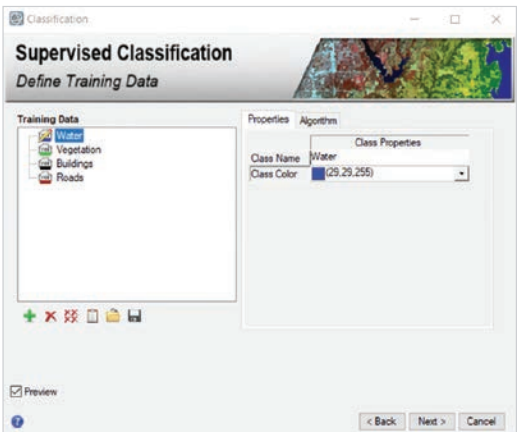
3. The multispectral file that you opened should be listed under Raster File. Click Next. The Classification Type panel appears.



ENVI provides a Classification Workflow that allows users to create a Landcover / Terrain Categorization (TERCAT) product. This process can be carried out by the software in an "Unsupervised" manner or by user-assigned spectra in a "Supervised" manner. The following exercise will conduct a Supervised classification.

4. Select Training Data to indicate that this is a 'Supervised' classification. Click Next. The Supervised Classification panel appears.

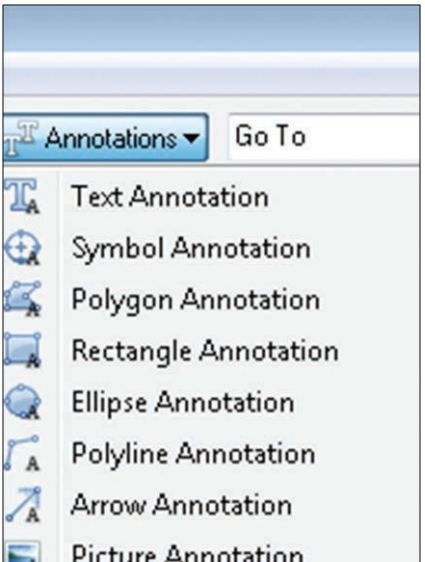
5. Click the Add Class button ✚ to add Training Data classification(s). Assign Class Names per region to the right under the Properties tab. Delete training data if necessary by highlighting items and clicking ✕.



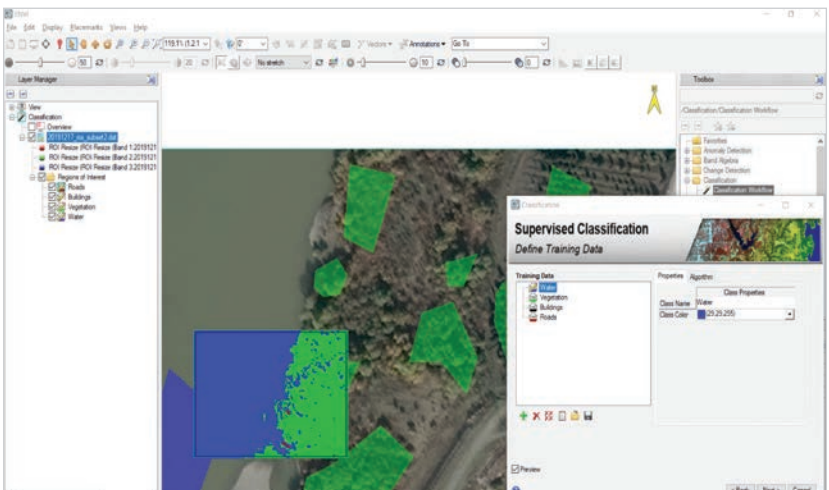
6. Once the desired classes have been created, highlight a single class at a time. Then select spectral samples for that class by drawing polygons in the image around the features of interest. Use the ROI drawing tool in tandem with the navigation tools to zoom and pan around.

7. The mouse wheel is useful in allowing you to zoom in and out based on the cursor position while in annotation mode.

8. Select 8 to 10 varying spectral samples for each class by drawing a polygon or rectangle. Then right-click and select Accept or Delete.



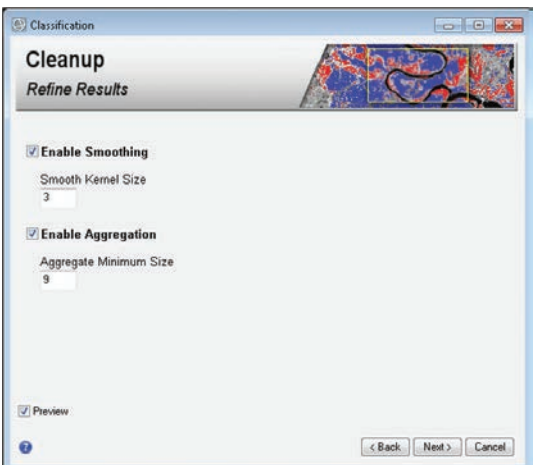
9. Once you are satisfied with spectral selection, toggle the Preview check box to launch a Portal View so you can inspect the results so far. Drag the Portal View around the View to inspect.



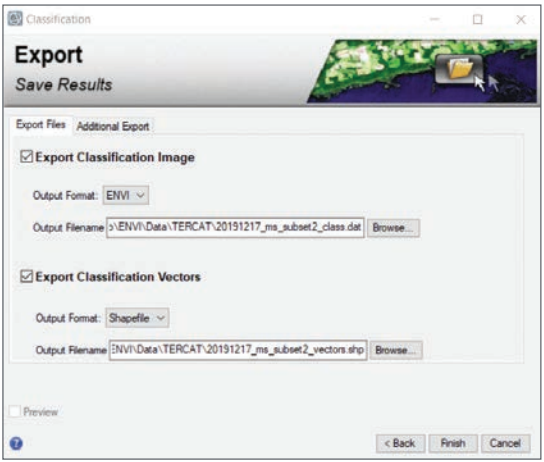
10. If you are not satisfied with the previewed result, you can click the icon in the ENVI toolbar and select specific ROIs to remove. Press the Delete key to remove them if needed. Or, draw more polygons around representative features.

11. Click Next. The Cleanup panel appears.

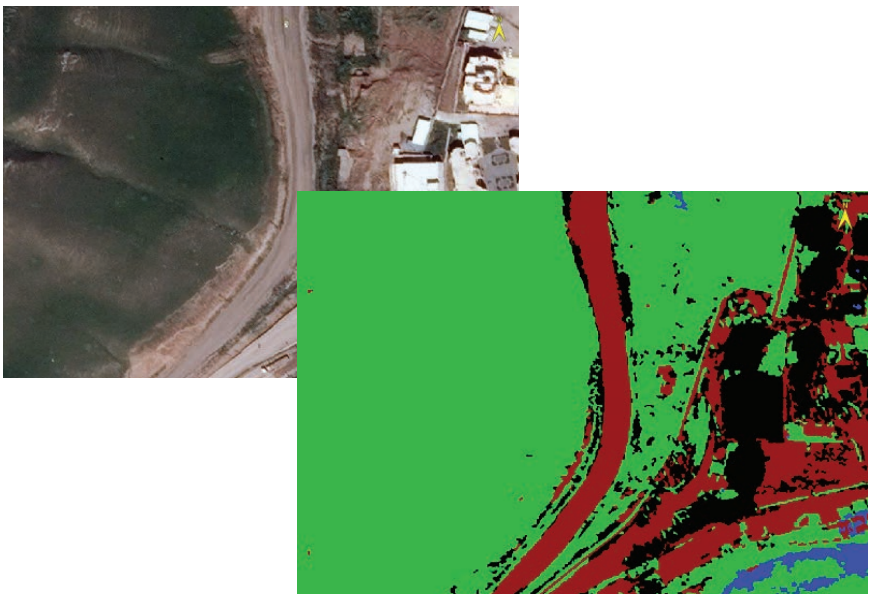
12. The Cleanup panel allows you to optionally use smoothing algorithms to generalize the classification results. Adjust or accept the defaults and click Next. The Export panel appears.



13. ENVI will automatically append the image and vector output files to the root name. ENVI will also save the output files in the same directory as the input files. You can accept these defaults or change them by clicking Browse. Click Finish.



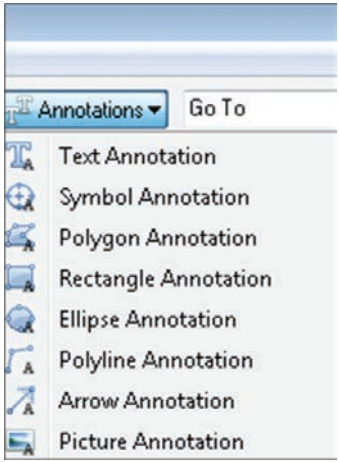
Input image, new image and vectors TERCAT examples:



EXPORT TO RPF

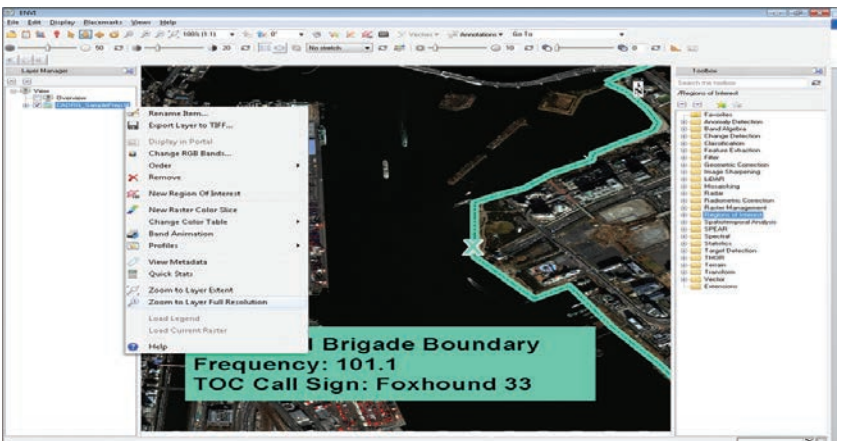
1. Generate a GEOTIFF with specialized vector overlays, drawings, pictures and/or text annotations. You can do this using the Annotations menu or in ArcMap as long as the GEOTIFF is accompanied by a corresponding World File.

2. In ENVI, open the newly created GEOTIFF by selecting File > Open and navigating to the location on disk.



ENVI enables creation of the following RPF/CADRG resolutions: (1: 5,000,000 GNC (754m), 1: 2,000,000 JNC (300m), 1: 1,000,000 ONC (150m), 1:500,000 TPC (75m), 1: 250,000 JOG (37.5m), 1: 100,000 TLM (15m) and 1: 50,000 TLM (7.5m). Quality will depend on the scale and dots per inch (DPI) of the input, so take this into account during data preparation.

3. Right-click on the file in the Layer Manager and select Zoom to Layer Full Resolution. Visually inspect the GEOTIFF. Next, right-click on the file in the Layer Manager and select Zoom to Layer Extent. If you are not satisfied with the visual inspection, make necessary adjustments such as DPI, overlays, etc., then export again before proceeding.

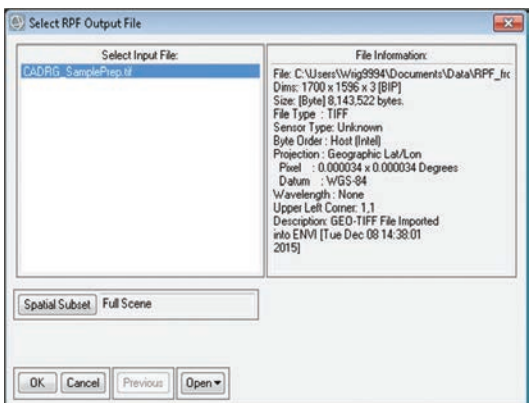


The GeoTIFF in this example was created in ArcMap at Scale 1: 25,000. It was exported from the Data view as a TIFF file with the Write World File option enabled under the General tab, and the Write GeoTiFF Tags option enabled under the Format tab. A DPI of 200 was used.

4. In ENVI, click File > Save As > CADRG once you are satisfied with the newly created GEOTIFF.

5. Highlight the GEOTIFF under Select Input File.

6. Click OK. The RPF (CADRG) Output Parameters dialog appears.



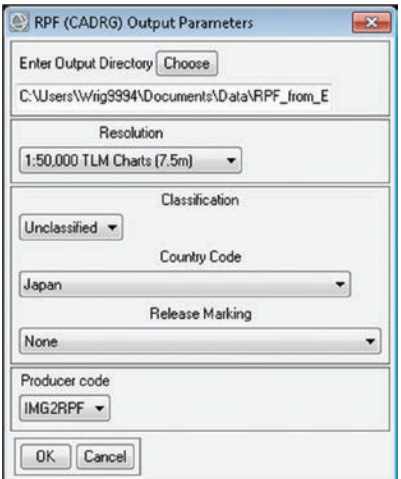
7. Select an Output Directory.

8. Set the desired Resolution from the drop-down menu. In this example, 1: 50,000 TLM Resolution was chosen.

9. Optional: Complete Classification, Country Code, and Release metadata.

10. Choose IMG2RPF as the Producer Code.

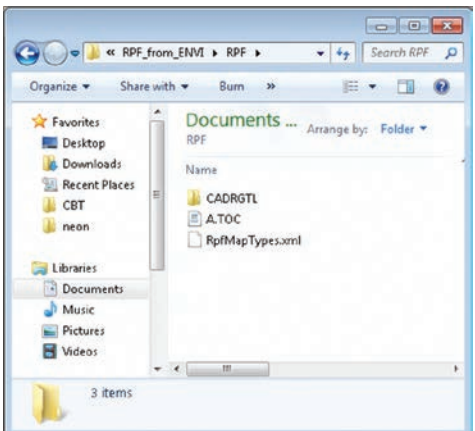
11. Click OK.



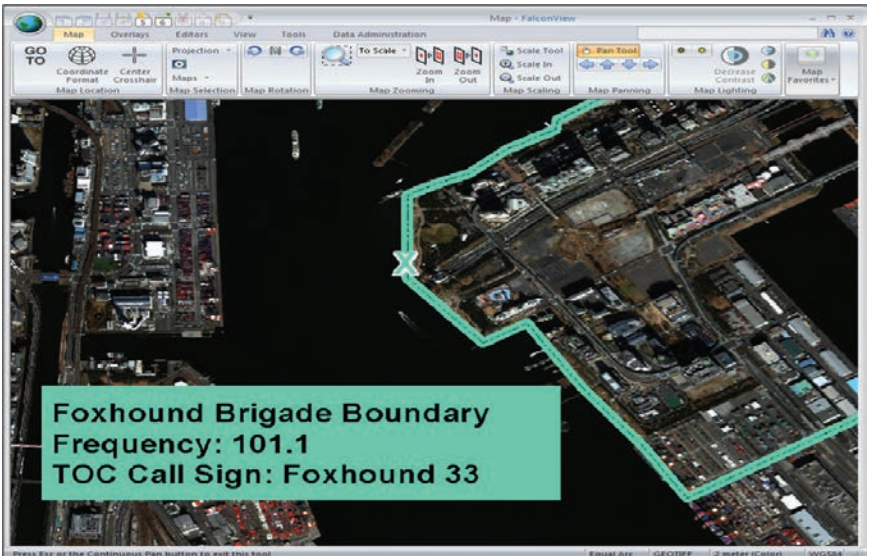
12. Once processing is complete, navigate to the output directory and ensure you have the RPF data and the A.TOC file.

13. Your Output directory's contents should look similar to the example on page 64.

14. Visually inspect the newly created RPF data using Falconview. If dissatisfied, make refinements and repeat the preceding steps. Once satisfied, burn the entire RPF folder—including the A.TOC file—to disk to be loaded onto military aircraft and/or ground vehicles that rely on RPF data.



Results displayed in Falconview:

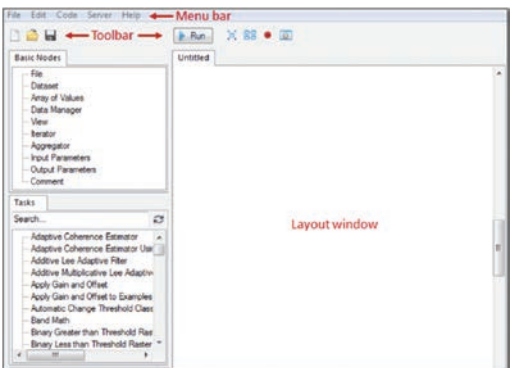


ENVI MODELER

Start the ENVI Modeler using one of these options:

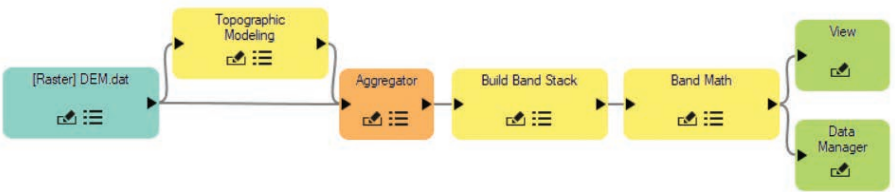
- Select Display > ENVI Modeler from the ENVI menu bar.
- In the ENVI Toolbox, expand the Task Processing folder and double-click ENVI Modeler.
- Drag and drop a .model file from Windows Explorer to the ENVI application.

This figure shows the main components of the ENVI Modeler user interface.



Elements of a Model

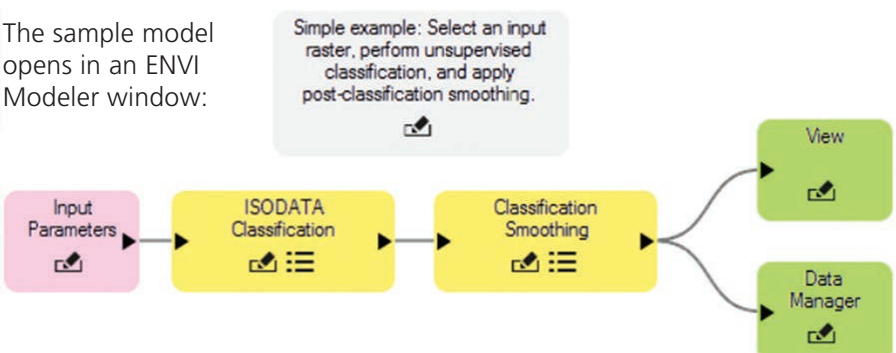
A model consists of two primary elements: nodes and connectors. A node is a basic building block such as an input file, task, or other operation. A connector is a grey line that connects nodes to one another via their input and output parameters. The following figure shows an example model with nodes and connectors:



1. Click the Open button in the ENVI menu bar. Navigate to the examples folder in your ENVI installation path, and select the file "Modeler_Tutorial.model." The default installation path is as follows, where xx is the ENVI version number:

    - Windows: C:\Program Files\Harris\ENVIxx\examples\
    - Linux: /usr/local/harris/envixx/examples/
    - Macintosh: /Applications/harris/envixx/examples/

The sample model opens in an ENVI Modeler window:



2. Click the Run button Run in the ENVI Modeler window to run the model.

3. The model stops at the Input Parameters node so that you can select an image. In the Input Parameters dialog, click the Browse button ....

4. In the Data Selection dialog, click the Open File button .

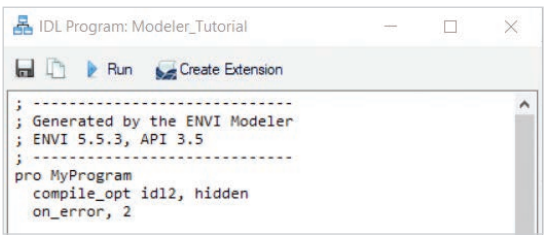
5. Select a multispectral file; for example, "qb_boulder_msi." This file is located in the data folder of the ENVI installation path. Click OK.

6. Click OK in the Input Parameters dialog. The model runs and displays a smoothed classification image derived from the input multispectral image. The classification image is also added to the Data Manager and the view.

Generate and Run Code From Models

IDL Programs and ENVI Toolbox Extensions

To generate IDL code for a model, select Code > Generate IDL Program from the ENVI Modeler menu bar. You must have an ENVI+IDL license to use this feature. The IDL code is displayed in a new Program window; for example:



From here, you can do the following:

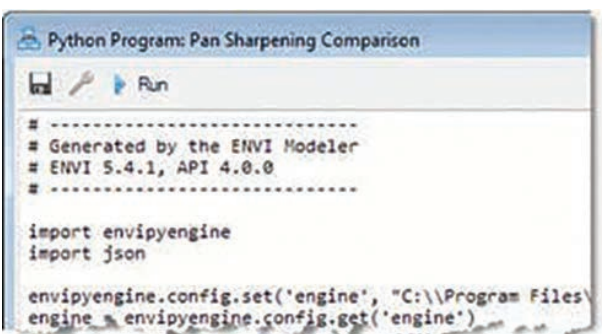
- To save the program to an IDL program file (.pro), click the Save button .
- To run the program, click the Run button Run.
- To create an extension, click the Create Extension button Create Extension.

Python Programs

You can generate Python code for a model if you want to use Python to call the ENVI Task Engine. To run the Python code that is generated, you must have the following installed:

- Python
- pip
- One of the following: ENVI Py Engine, ENVI Py for ArGIS® Pro, or ENVIPy for ArcMap.

Select Code > Generate Python Program from the ENVI Modeler menu bar. The Python code is displayed in a new Program window; for example:



From here, you can do the following:

- To save the program to a Python program file (.py), click the Save button .
- To run the program, click the Run button Run .
- To configure the output directory of the Python program, click the Configure Task Engine button . Enter an Output Directory and click OK. All of the files generated by the model will be written to that directory.

When running a Python Program, the results will not be displayed or added to the Data Manager in ENVI. All the output rasters or vectors that are connected to an Output Parameters node will be preserved in the output folder. Other immediate files that are created will be immediately deleted upon running the program.

Metatasks

You can save a model to a task file (.task) on disk so that you share the task with others or use it in the ENVI API, similar to a custom task. Follow these steps to save a model as a task file:

1. Decide what inputs and outputs your metatask needs to expose. See Define Metatask Inputs and Outputs in the Create Models topic for details.

2. Select Code > Generate Metatask from the ENVI Modeler menu bar. The task code is displayed in a new Metatask dialog.

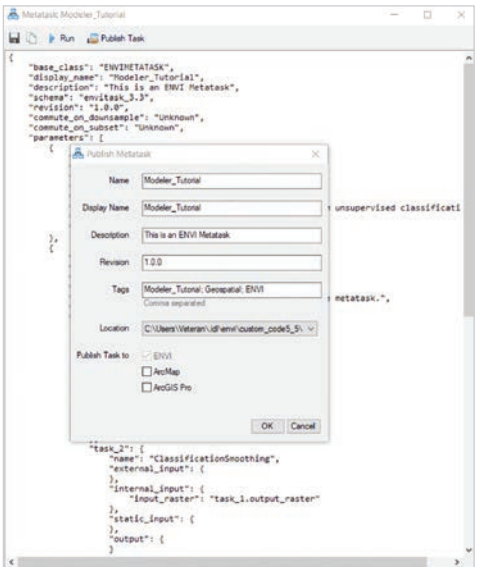
3. Choose a save option:

   - Click the Save button in the Metatask dialog. Select an output filename and directory for the .task file. See "Deploy Custom Task Files" in the ENVI documentation for recommendations on where to save .task files on your system.

   - Click the Publish Task button in the Metatask dialog. In the Publish Task dialog, enter a Name, Display Name, Description, Revision, and Tags for the task.

- The default location for saving the task is shown. If you want to change this, select another option from the dropdown list.

- If you want to publish the task to ArcMap or ArcGIS Pro, check the appropriate box; this will make the task available to use directly from the ArcMap and/or ArcGIS Pro system toolbox the next time the product is started. Click OK.



ANALYTICS, INSIGHTS, ANSWERS YOU CAN TRUST.

FOR MORE INFORMATION:

L3HarrisGeospatial.com
Email: GeospatialInfo@L3Harris.com
Phone: 303.786.9900

L3Harris Technologies is an agile global aerospace and defense technology innovator, delivering end-to-end solutions that meet customers' mission-critical needs. The company provides advanced defense and commercial technologies across air, land, sea, space and cyber domains.

L3HARRIS™
FAST. FORWARD.

385 Interlocken Crescent, Ste 300
Broomfield, CO 80021
t 303 786 9900
GeospatialInfo@L3Harris.com